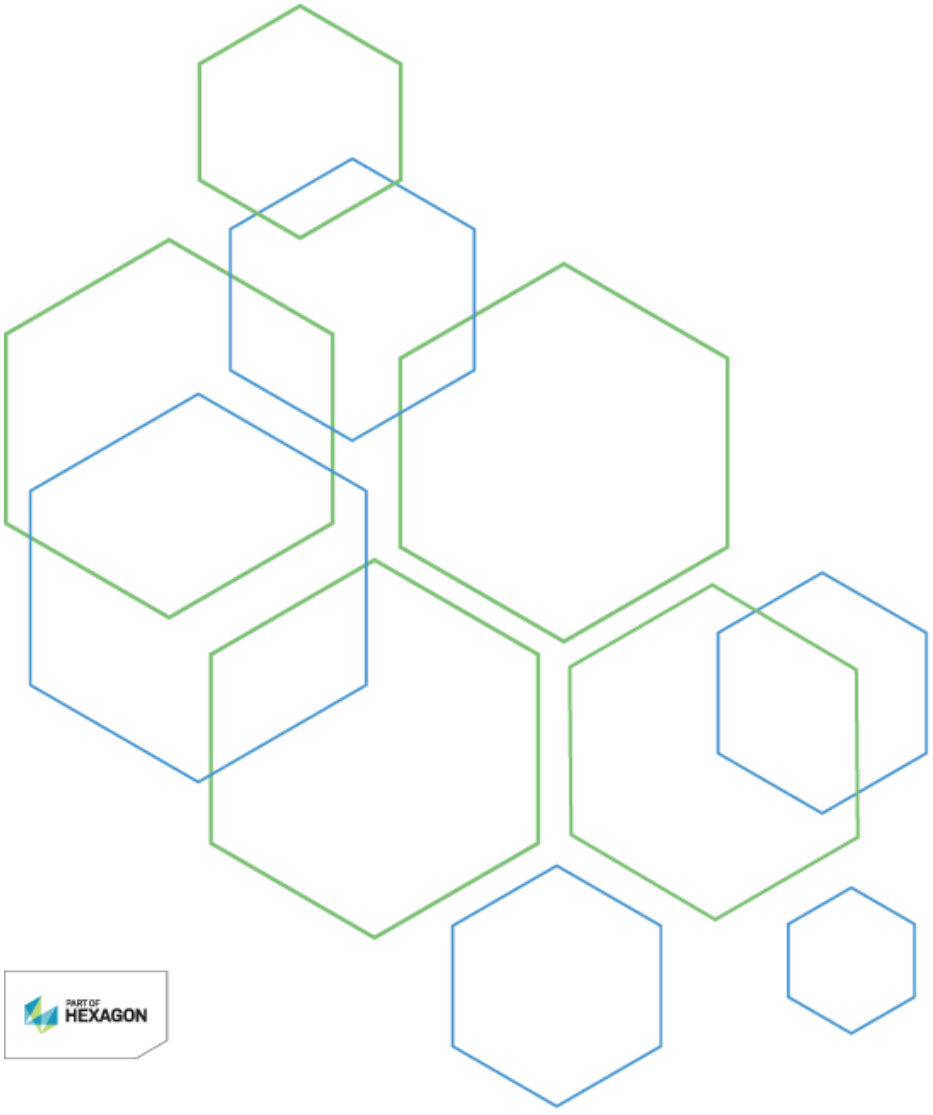
INTERGRAPH Smart 3D

Structure

User's Guide



PART OF HEXAGON

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November 2016

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Preface

This document is a user's guide for the Structure functionality of Intergraph Smart™ 3D and provides command reference information and procedural instructions.

Documentation Comments

For the latest support information for this product, comments or suggestions about this documentation, and documentation updates for supported software versions, please visit *Intergraph Smart Support* (*https://smartsupport.intergraph.com*).

What's New in Structure

The following changes have been made to the Structure task.

Version 2016 (11.0)

- The new **Member Autoconnect** (on page 488) command creates frame connections between existing members in the model automatically using a predefined set of rules and tolerances. When you import, move, copy and paste with a move, or mirror copy members in the model, the frame connections are set as unsupported. But now with automatic connection, the software finds the needed frame connections for you. (P2 CP:255984; P2 CP:267994)
- You can now place built-up members, including tapered members, in a plant model. Before you were limited to placing built-up members only in a marine model. (P2 CP:136769)
- When placing members, the active coordinate system defined in the PinPoint ribbon is honored. For example, this allows you to place columns with a 0-degree beta angle in a rotated coordinate system and have the columns align correctly to the rotated coordinate system. This feature is available in **Place Linear Member Systems** (on page 35), **Place Bracing** (on page 113), **Place Framing Members** (on page 127), and **Place Columns at Grid Intersections** (on page 138). In **Place Vessel Supports** (on page 142), the X-, Y-coordinates of the selected equipment displays in the active coordinate system. (P2 CP:19956)
- You can now export marine plate objects to a CIS/2 file. (P2 CP:253427)
- You can now import bocad® models into Smart 3D. For more information, see *Importing and Exporting Structure* (on page 507) and *Import bocad* (on page 526). (P2 CP:250191)
- When you convert handrails to members, the handrail posts are now automatically connected to nearby structural objects with either the Axis-Along or Surface-Default frame connections. This allows you to place assembly connections on the handrail posts to obtain weld information and manufacture any base plates or attachments that are needed for the connection. (P2 CP:250589)
- Added descriptions of molded conventions for the individual plates of a designed (built-up) member. For more information, see *Molded Conventions Tab (Nonlinear Plate System Properties Dialog Box - Designed Member)* (on page 89). (P2 CP:264474)
- Added a new locate filter, **Construction Graphics**. For more information, see *Selecting Objects* (on page 19). (P2 CP:271166)

- You can specify a new sketching plane when you edit an opening with a missing sketching plane. For more information, see **Place Opening** (on page 367). (P2 CP:279815)
- You can now use the measure command to measure from fireproofing geometry to objects in the model to check for clearance. (P2 CP:155070)
- Tapered and built-up members can now be exported to a physical CIMsteel file. (P3 CP:229011)
- Curved members can now be assigned **Piecemark** and **Assembly Mark** properties. (P3 CP:228984)
- You can now import CIS/2 files without having the software compare the GUIDs in the file with the GUIDs in the model. This ability to import and ignore IDs allows you to import the same CIS/2 file multiple times into the same model to create a unique copy each time the file is imported. (P3 CP:249402)
- The **Place Bracing** (on page 113), **Place Framing Members** (on page 127), **Place Columns at Grid Intersections** (on page 138), and **Place Vessel Supports** (on page 142) commands now support the use of built-up sections. (P3 CP:147163)
- For piping and equipment objects in plant mode, **Insulation**, **Equipment Hole**, **Maintenance**, and **Operation** aspects displaying in the 3D graphic view also display in the Sketch 2D environment. For more information, see *Format View Dialog Box* in the *Common User's Guide*, **Place Opening** (on page 367), and **Place Slabs** (on page 220). (P4 CP: 119768)

SECTION 1

Structure

The **Structure** task places and modifies structural objects. Using this task, you can place beams, columns, braces, truss elements, cables, equipment foundations, column footings, openings, slabs, walls, and connections in your model. You also can create custom section shapes using 2D Symbols and place those custom sections in the model.

The **Structure** task also provides for traffic needs by placing stairs, ladders, and handrails.

NOTE Although not required, we recommend that you place grids using the **Grids** task before placing structural members.

The **Structure** task has these commands:



Select - Used to select objects in the model. For more information, see *Selecting Objects* (on page 19).



Place Linear Member System - Places columns, beams, braces, and other linear members in the model. For more information, see *Place Linear Member Systems* (on page 35).



Place Curve Member - Places a curved column, beam, or brace member in the model. For more information, see *Place Curve Members* (on page 109).



Place X Bracing - Places vertical cross bracing or vertical chevron bracing in the model. For more information, see *Place Bracing* (on page 113).



Place Framing Members - Places secondary framing members between two supporting members that you specify. For more information, see *Place Framing Members* (on page 127).



Place Columns at Grid Intersections - Places columns at grid intersections. For more information, see *Place Columns at Grid Intersections* (on page 138).



Place Vessel Supports - Places structural members that support vertical vessels directly (typically four lugs that rest on the members) or that
support the grating around the vessel. For more information, see *Place Vessel Supports* (on page 142).



Place Can - Places a can in the model. A can is a reinforcing connection piece on a member to increase the surface area available for connections or to provide increased wall thickness or both. For more information, see *Place Can* (on page 148).



Place Assembly Connection - Places assembly connections between linear member systems. For more information, see *Place Assembly Connection* (on page 174).



Place Split - Divides a member system into multiple member parts while maintaining the design intent of the original member system. For more information, see *Place Splits* (on page 191).



Trim Member - Manually copes, snips, and planar trims member ends. For more information, see *Trim Members* (on page 201).



Place Fireproofing - Places fireproofing on selected members. For more information, see *Place Fireproofing* (on page 207).



Place Slab - Places slabs, plates, and grates in the model. For more information, see *Place Slabs* (on page 220).



Place Wall - Places a wall in the model. For more information, see *Place Walls* (on page 244).



Toggle Wall Corner - When two separate walls join at a corner, one wall extends past the other wall to form the corner. You can use this command to toggle which wall extends past the other. For more information, see *Toggle Wall Corner* (on page 277).



Place Equipment - Places equipment in the model. In the **Structure** task, use this command to place doors and windows. For more information, see *Place Equipment* (on page 278).



Place Designed Equipment - Places equipment types that have been defined in the reference data. For more information, see *Place Designed Equipment* (on page 298).



Place Designed Equipment Component - Places equipment components that have been defined in the reference data. For more information, see *Place Designed Equipment Component* (on page 314).



Place Designed Solid - Creates customized solids and equipment using shapes that add material to or subtract material from the designed solid. For more information, see *Place Designed Solid* (on page 317).



Place Shape - Adds additional shapes or equipment objects to an existing designed equipment type. The icon displayed on the toolbar is the icon of the last shape selected from the palette. For more information, see *Place Shape* (on page 328).



Place Imported Shape from File - Adds geometry to a designed equipment object that was modeled with solid modeling software and saved to a format. For more information, see *Place Imported Shape from File* (on page 363).



Place Opening - Places openings (holes) in slabs, walls, and linear member systems. For more information, see *Place Opening* (on page 367).



Place Stair - Places stairs in the model. For more information, see *Place Stairs* (on page 383).



Place Ladder - Places ladders in the model. For more information, see *Place Ladders* (on page 398).



Place Handrail - Places a handrail in the model following a path that you specify. For more information, see *Place Handrails* (on page 418).



Place Handrail by Member - Places a handrail on selected members without having to define a path. For more information, see *Place Handrails by Member* (on page 458).



Place Footing - Places a column footing in the model. For more information, see *Place Footings* (on page 463).



Place Equipment Foundation - Places a foundation for selected pieces of equipment. For more information, see *Place Equipment Foundations* (on page 479).



Member Autoconnect - Connects existing built-up or standard members using a pre-defined set of rules and tolerances. For more information, see *Member Autoconnect* (on page 488).

Import Structure - Imports a CIS/2 file into the model. This command is on the **File** menu. For more information, see *Import Structure* (on page 516).

Export Structure - Exports the structural physical model to a CIS/2 or IFC file. This command is on the **File** menu. For more information, see *Export Structure* (on page 512).

New Mapping File - Creates an XML mapping file for the section and material names used in the software and third-party application. This command is on the **File** menu. For more information, see *New Mapping File* (on page 535).

SECTION 2

Structure Workflow

All structure objects are placed in the model using information defined in the structure reference data. Your first step should be to review, edit, and otherwise customize the delivered structure reference data using the **Catalog** task. Refer to the Catalog documentation. If you prefer, you can still review, edit, and create structure reference data using Microsoft Excel workbooks. Refer to the *Structure Reference Data Guide* for more information on editing reference data using workbooks.

After the reference data is customized to suit your needs, consider going to the **Systems and Specifications** task and defining the systems that you want in your model. While not absolutely required that you create your systems first, doing so keeps you from having to edit your structural objects after placement to assign them to the correct system. In addition, we recommend that you create your elevations and grids using the **Grids** task before placing structural objects.

After the structure reference data and the needed systems and grids are defined, you can begin placing structural objects in your model.

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Structure Common Tasks

The following tasks are used frequently in the **Structure** task.

Customize Reference Data

Create new member types by editing the **AllCodeLists.xls** workbook. For more information, see the *Structural Reference Data Guide*.

Review, add, and edit material information. For more information, see the *Structural Reference Data Guide*.

Place Members

Place columns in the model. For more information, see *Place Members using Discrete Placement* (on page 54).

Place beams in the model. For more information, see *Place Members using Contiguous Placement* (on page 55).

Places braces in the model. For more information, see *Place Vertical Cross Bracing* (on page 117), *Place Vertical Chevron Bracing* (on page 121), and *Position Framing Members at Equal Spacing* (on page 135).

Split member systems into member part as needed to resolve interferences, place column splices, and other modeling issues. For more information, see *Split a Member that Intersects another Member* (on page 195) and *Split Columns at a Plane* (on page 197).

Place assembly connections in the model. For more information, see *Place an assembly connection* (on page 181).

Place Slabs

Place slabs in the model. The software provides great flexibility in placing slabs. For more information, see:

Place a Slab by Selecting Boundary Objects (on page 226)
Place a Slab by Drawing 2-D Boundaries (on page 228)
Place a Slab by Drawing 3-D Boundaries (on page 230)
Place a Slab by Selecting Objects and Drawing 2-D Boundaries (on page 230)
Place a Slab by Selecting Objects and Drawing 3-D Boundaries (on page 231)

Place Walls

Place walls in the model. The software provides great flexibility in placing walls. For more information, see:

Place a Wall by 2-D Sketch (on page 248)
Place a Wall by 3-D Sketch (on page 261)

Place Openings

Place openings in slabs, walls, and members in the model. You can define openings by boundaries, by drawing the opening, or by selecting a shape from the catalog for the opening. You can place doors and windows by selecting a door or window from the catalog.

For more information on placing an opening by boundaries, see *Place an Opening by Boundaries* (on page 371).

For more information on placing an opening by drawing the opening, see *Place an Opening by Drawing* (on page 374).

For more information on placing an opening by using a shape from the catalog, see *Place an Opening by Shape* (on page 372).

For more information on placing doors, see *Place Doors from the Catalog* (on page 284).

For more information on placing windows, see *Place Windows from the Catalog* (on page 285).

Model Foundations

Place equipment foundations in your model. For more information, see *Place an Equipment Foundation (By Equipment)* (on page 481).

Place column footings in your model. For more information, see *Place a Single Footing* (on page 466) or *Place a Combined Footing* (on page 469).

Provide for Traffic

Place stairs, or incline ladders, in your model. For more information, see *Place a Stair* (on page 387).

Place ladders in your model. For more information, see *Place a Ladder* (on page 401).

Place handrails in your model. For more information, see *Place a Handrail* (on page 420) and *Place a Handrail by Member* (on page 461).

Exporting the Physical Model

When all the primary and secondary members are modeled, you may want to export the physical model to a detailing application. For more information, see *Export Structure Model* (on page 512).

Selecting Objects

All objects in the **Structure** task have properties that you can edit. Using the **Select** command on the vertical toolbar, you select the object that you want to edit.



An important part of the **Select** command is the **Locate Filter** box that appears on the ribbon. The **Locate Filter** box contains the available, predefined filters for the **Select** command. When you choose a filter in the **Locate Filter** box, the software allows you to select only the filtered objects in a graphic view and in the **Workspace Explorer**. For example, if you select **Member Systems**, you can select only member systems in a graphic view or in the **Workspace Explorer**.

The **Structure** task includes these filters:

Assembly Components

Limits your selection in a graphic view or in the **Workspace Explorer** to assembly components placed by an assembly connection or by an assembly footing or foundation.

Cans

Limits your selection in a graphic view or in the **Workspace Explorer** to Cans. You can place Can using *Place Can* (on page 148) (only available in marine mode).

Construction Graphics

Limits the selection of items to construction graphics.

Control Points

Limits your selection in a graphic view to control points.

Designed Members

Limits your selection in a graphic view or in the **Workspace Explorer** to designed members, which are non-standard members comprised from a collection of individual plates.

Equipment

Limits your selection in a graphic view or in the **Workspace Explorer** to equipment. Remember that doors and windows are considered pieces of equipment. You can place doors and windows using **Place Equipment**. For more information, see *Place Equipment* (on page 278).

Equipment Components

Limits your selection in a graphic view or in the **Workspace Explorer** to equipment components.

Equipment Foundations

Limits your selection in a graphic view or in the **Workspace Explorer** to equipment foundations. You can place equipment foundations using **Place Equipment Foundation**  . For more information, see *Place Equipment Foundations* (on page 479).

Footings

Allows you to select footings in a graphic view or in the **Workspace Explorer**. You can place footings using **Place Footing**  . For more information, see *Place Footings* (on page 463).

Frame Connections

Limits your selection in a graphic view or in the **Workspace Explorer** to frame connections. For more information about these connections, see *Frame Connections* (on page 24).

Handrails

Limits your selection in a graphic view or in the **Workspace Explorer** to handrails. You can place handrails using **Place Handrail**  or **Place Handrail by Member**  . For more information, see *Place Handrails* (on page 418) or *Place Handrails by Member* (on page 458).

Insulation

Limits your selection in a graphic view or in the **Workspace Explorer** to fireproofing insulation. You can place fireproofing using **Place Fireproofing**  . For more information, see *Place Fireproofing* (on page 207).

Ladders

Limits your selection in a graphic view or in the **Workspace Explorer** to ladders. You can place ladders using **Place Ladder**  . For more information, see *Place Ladders* (on page 398).

Member Assembly Connections

Limits your selection in a graphic view or in the **Workspace Explorer** to member assembly connections. For more information about these connections, see *Member Assembly Connections* (on page 29) and *Place Assembly Connection* (on page 174).

Member Parts

Limits your selection in a graphic view or in the **Workspace Explorer** to member parts. One or more member parts define a member system.

TIP If you not sure whether to use **Member Systems** or **Member Parts** when you want to edit properties, try **Member Parts** first.

Member Systems

Limits your selection in a graphic view or in the **Workspace Explorer** to member systems. A member system contains one or more member parts. You can place member systems using **Place Linear Member**  or **Place Curved Member**  . For more information, see *Place Linear Member Systems* (on page 35) and *Place Curve Members* (on page 109).

Openings

Limits your selection in a graphic view or in the **Workspace Explorer** to openings (holes). You can place openings using **Place Opening**  . For more information, see *Place*

Opening (on page 367).

Shapes

Limits your selection in a graphic view or in the **Workspace Explorer** to shapes.

Slab Assembly Connections

Limits your selection in a graphic view or in the **Workspace Explorer** to slab assembly connections. For more information about these connections, see *Slab Assembly Connections* (on page 33).

Slabs

Limits your selection in a graphic view or in the **Workspace Explorer** to slabs. You can place slabs using **Place Slab** . For more information, see *Place Slabs* (on page 220).

Solids

Limits your selection in a graphic view or in the **Workspace Explorer** to solids.

Split Connections

Limits your selection in a graphic view or in the **Workspace Explorer** to split connections. For more information about these connections, see *Place Splits* (on page 191).

Stairs

Limits your selection in a graphic view or in the **Workspace Explorer** to stairs (incline ladders). You can place stairs using **Place Stair** . For more information, see *Place Stairs* (on page 383).

Structure

Allows you to select any object in a graphic view or in the **Workspace Explorer** that was placed using the **Structure** task. Objects placed using other tasks, such as equipment, cannot be selected using this filter.

Wall Assembly Connections

Limits your selection in a graphic view or in the **Workspace Explorer** to wall assembly connections.

Wall Parts

Limits your selection in a graphic view or in the **Workspace Explorer** to wall parts.

Wall Systems

Limits your selection in a graphic view or in the **Workspace Explorer** to wall systems. A wall system contains one or more wall parts. You can place wall systems using **Place Wall** . For more information, see *Place Walls* (on page 244).

All

Allows you to select any object, even objects created in another task.

 Use the **Inside** fence command to select all objects entirely inside the fence.

 Use the **Inside/Overlapping** fence command to select all objects entirely inside the fence and those objects outside but touching the fence at some point.

Members

You can place linear and curved members. All linear members are placed using *Place Linear Member Systems* (on page 35)  . All curved members are placed using *Place Curve Members* (on page 109)  . Both commands provide options for selecting member type category, type, section name, cardinal point, and other options during placement.

In addition, the software provides several member placement productivity commands in the **Structure** task that, depending on what you are doing, might be a better choice than **Place Linear Member System**. For example, to place a column at each grid intersection in one operation, use *Place Columns at Grid Intersections* (on page 138)  . If you want to place cross bracing, use *Place Bracing* (on page 113)  . Use *Place Framing Members* (on page 127)  to place secondary framing members in a bay. If you want to place support members around a vertical vessel, use *Place Vessel Supports* (on page 142)  . Before you start placing members however, there are concepts that you need to know.

NOTE Although not required, we recommend that you place grid planes, elevations planes, and grid lines using the **Grids** task before placing structural members.

Standard Member Systems

Member systems are logical collections of member parts that maintain the design basis and physical alignment of the member parts for analysis, design, and manufacturing. For example, in vertical cross-bracing, typically one of the vertical braces is split into two parts so that it does not interfere with the other vertical brace in the cross. The member system for that split vertical brace maintains the co-linear alignment of the two parts when you move either outside corner of the vertical brace. Another example of a member system would be a jacket leg. The leg is comprised of different parts, including cans that have different cross-section sizes, but you want the entire leg to move as a single member. Use *Place Splits* (on page 191)  to split member systems into member parts.

Member systems connect to other member systems using Frame Connections. For more information, see *Frame Connections* (on page 24).

Member Parts

Member parts are children of a standard member system and represent the physical parts in the model. Member parts connect logically to other member parts using assembly connections. For more information, see *Member Assembly Connections* (on page 29).

Designed Members

In marine mode and material handling mode, the software places a designed member, composed of individual plates, in the same fashion as a standard rolled member. For example, a designed member with a W-section has a plate for the top flange, a plate for the web, and a plate for the bottom flange. These plates behave just like all other plates in the model, which means you can detail and manufacture the plates in the **Structural Detailing** and **Structural Manufacturing** tasks. You can even edit the properties of the individual plates to adjust the plate thickness, for example, without affecting the designed member's cross-section. It is important to know that if a designed member is placed in a lapped configuration (for example, as a plate stiffener) on a plate system, no logical connections are created along the designed member’s toe trace. If needed, the logical connections must be placed manually.

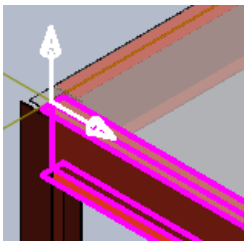
Because designed members are composed of plates, you can place members that are prismatic, tapered, and non-prismatic in the model. Sections for designed members are delivered in two section libraries: **Built-Up-English** and **Built-Up-Metric**. Use designed members to place tapered, can, cone, haunch, and other specialty members in your model.

Member Type Category and Type

Member categories are broader groupings of member types. For example, the software delivers a member type category called column. In the column type category are member types called column and stud. When placing a member, you have to select a member type category and a member type. You can define your own member categories and member types by editing the **Structural Member Type** select list in the **Catalog** task. Refer to the *Catalog User's Guide* for more information.

Member Local Coordinate System and Orientation

The software uses the following convention to determine the local coordinate system of a member. The member's local x-axis is along the member axis from member start to member end. The member's local z-axis is the strong axis of the member cross-section. The member's local y-axis is determined by the right- hand-rule using the local x- and z-axes.



When placing members, the software sets the local z-axis of the member parallel to the global Z-axis by default. However, if you rotate the member such that the local x-axis of the member becomes parallel to the global Z-axis, then the software switches the local z-axis of the member to be parallel to the positive global X-axis. When you select a member part, the software indicates the member's x- and z-axes with arrows that display at the member part start end.

Discrete or Contiguous Placement Methods

When placing members in the model, you can use either discrete placement or contiguous placement. The discrete placement method requires you to define both the start and the end points of the member, which is useful when placing columns. The contiguous placement method uses the end point of the previously placed member as the start point of the next member. This method is useful when placing multiple beams, because in most cases the end of the previous beam is where you want to start the next beam.

The **Start** and **End** commands on the ribbon are used to toggle between placement methods.

Frame Connections

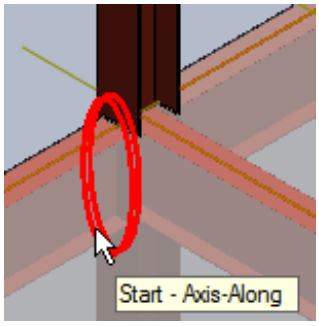
Frame connections describe the positioning relationship between member systems. This positioning relationship defines the member orientation and offset of the supported member in relation to the supporting member. Two frame connections are placed when you place the member system, one at each end. Because frame connections define relationships between member systems, the frame connection might prevent you from moving a member. For help in moving members, see *Move One End of a Member* (on page 60) and *Move a Member* (on page 60). Refer to *Members* (on page 22) for important related information about member systems. The types of frame connections are:

Align-Default/Lapped

An align frame connection places the end of the supported member system at the intersection of three surfaces that you define. You can define offsets from all three surfaces. This frame connection is useful, for example, when you want a column to move with a wall or bulkhead on a lower floor or deck.

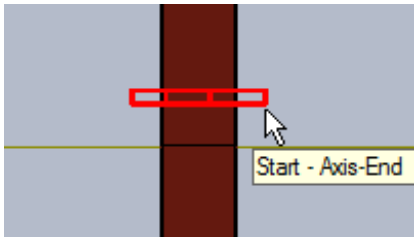
Axis Along

An axis along frame connection aligns the cardinal point on the supported member system with the cardinal point on the supporting member system. Use this frame connection when the member systems are different types (a beam framing into a column for example). Using this frame connection, the beam will slide along the length of the column, but will not cause the column to lengthen or shorten. You can specify an optional offset in all three directions.



Axis End

An axis end frame connection aligns the cardinal point on the supported member system with the cardinal point on the supporting member system. Use this frame connection when both member systems are of the same type (both members are columns, or both members are beams). If you move one member system end, this frame connection automatically moves the other member system end to maintain the connection. You can specify an optional offset in all three directions. Use this frame connection for column splices.

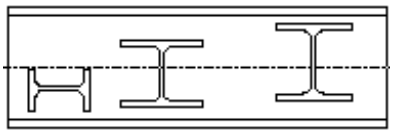


★ IMPORTANT The Axis End frame connection makes the members mutually editable. For example, if you move one member, the other member will extend or shorten to maintain

the connection. Because of this, if you change the permission group of one member, the software automatically changes the permission group of the other member to match.

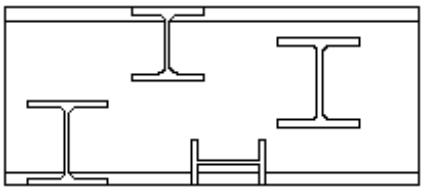
Centerline

A centerline frame connection uses the supporting member's centerline to position the supported member.



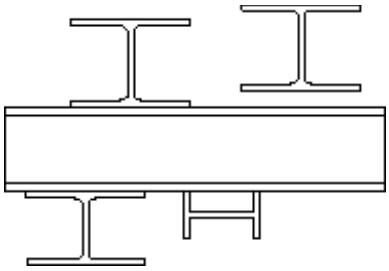
Flush

A flush frame connection uses the supporting member's top and bottom extent to position the supported member. The supported member typically lies within the body of the supporting member.



Seated

A seated frame connection uses the supporting member's top or bottom extent to position the supported member. The supported member typically rests against the supporting member, but can be offset.



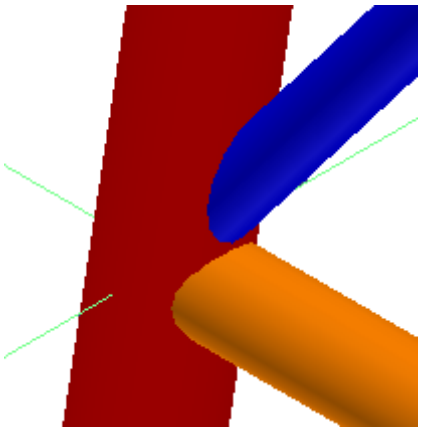
Gap

A gap connection defines offsets between members to provide clearance for welding or simply as a work point adjustment. Smart 3D can calculate the offset either axially along or radially around the support member. You must use the **More...** option and select the gap frame connection that you want to use. The software will not pick a gap frame connection when using the **By Rule** connection option.

There are three members in a gap frame connection:

- The *target* member is the member always to move and is the owner of the frame connection. The target member is shown as blue in the figure below.
- The *primary* member is the member to which the other two members are attached. The primary member is shown as red in the figure below.

- The *secondary* member is the third member in the joint. The software does not require the secondary member to be in the same plane as the target member. The secondary member is shown as orange in the figure below.



There are four gap frame connections available:

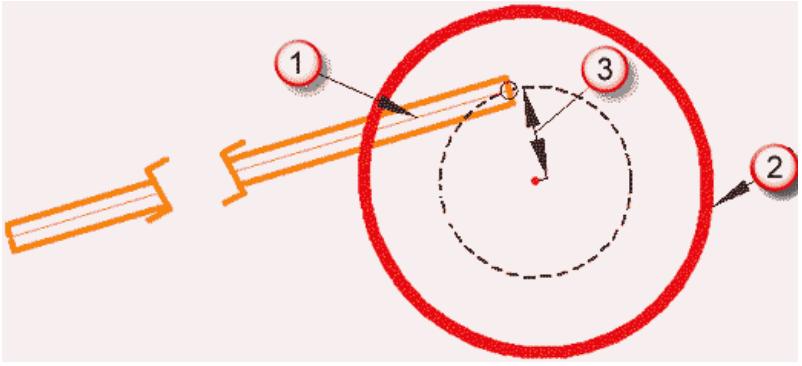
- **Radial Gap Single** - Moves the target member radial around the primary member. The gap is measured between the target and the secondary member.
- **Radial Gap Both** - Moves two target members radial around the primary member. The gap is measured between the two target members. Both target members' frame connection on that member end must be Radial Gap Both.
- **Axial Gap Single** - Move the target member axial along the primary member. The gap is measured between the target member and the secondary member.
- **Axial Gap Both** - Moves two target members axial along a primary member. The gap is measured between the two target members. Both target members' frame connection on that member end must be Axial Gap Both. For example, for chevron bracing you must define the Axial Gap Both frame connection for the ends of both braces for which you want define the gap. Then edit the properties of the Axial Gap Both frame connection to define the needed gap distance. The software recognizes the other brace frame connection is also an Axial Gap Both frame connection and moves both brace ends half the defined gap distance to achieve the gap.

Surface

A surface connection specifies the relationship between a supported member and the surface of the supporting object.

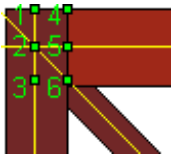
Tangent

A tangent connection specifies a relationship between a supported member (**1**) and a supporting member (**2**). A circle, whose radius you define (**3**), is projected out from the centerline of the supporting member. The supported member is then made tangent to that circle. You can also control the plane of the tangent circle.



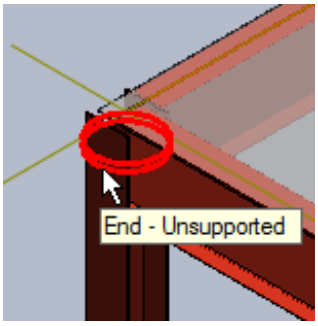
Vertical Corner Brace

A vertical corner brace connection specifies the location of a vertical brace that frames into a column-beam corner. You can define offsets in the X, Y, and Z-directions, and there are six work points to select from when using this connection.



Unsupported

An unsupported connection allows you to place a member end in free space. The top ends of columns have the unsupported frame connection.



Member Associativity

Members connected to other members using the Axis-Along, Seated, Centerline, Flush, and Tangent frame connections use the **Ratio** option for **Position Rule** to maintain the position relationship between the supported and supporting members.

A Surface frame connection maintains the current direction of the member if the opposite member end (regardless of the frame connection at that end) is moved.

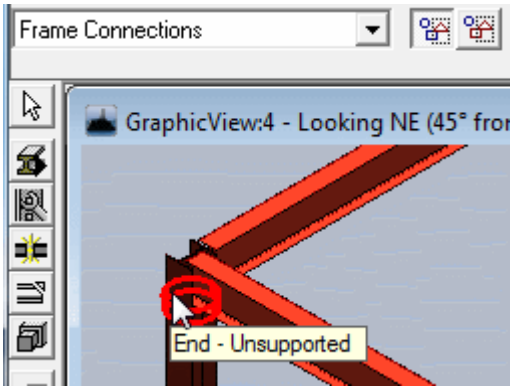
Selection of Frame Connections

During the placement of linear members, you can have the software determine frame connections by selecting the **By Rule** option. The software uses these rules based on supported member type category, type, permission group, and geometry as it connects to the supporting member to select frame connections:

- When the member connects to non-member objects, the software selects the "Unsupported" frame connection unless the non-member object is a surface, in which case **Surface-Default** is chosen.
- When a member connects to a single member, the software selects **Axis Along** unless:
  - The two members are end-matched and either the member being placed is a Brace member type or the member types and categories are the same, then the software selects **Axis End**.
  - The member being placed has a member type of Girt or Purlin and the two members are not parallel, then the software selects **Seated-Top**.
- When placing a member and you select another member's frame connection as the end point, the software reads both the frame connection's member and its optional supporting member. If the member being placed is coplanar with those two members, then the software selects **Vertical Corner Brace- WP2**.
- When placing a member and you select a split connection as the end point, the software reads the two members related to the split connection. If the member being placed is coplanar with those two members, then the software selects **Vertical Corner Brace-WP2**.

Locating Frame Connections

Frame connections do not display in the model except during member placement. However, if you set the **Locate Filter** to **Frame Connections**, you can locate and select frame connections for review and editing. Frame connections are located near the ends of member systems and appear as circles when you move the cursor over them.



When you select a frame connection, the software displays the frame connection type in the ribbon. Select the **Edit > Properties** command to edit the frame connection properties. You cannot copy a frame connection using the **Edit > Copy** command nor can you use a frame connection to measure distance using the **Tools > Measure** command.

See Also

Members (on page 22)
Edit a Frame Connection (on page 57)

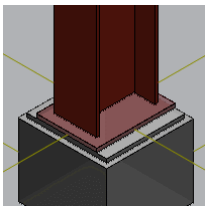
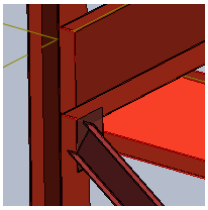
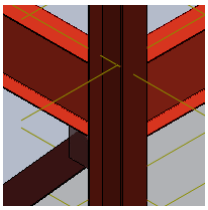
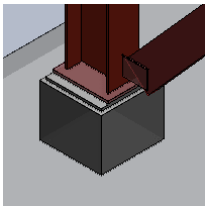
Member Assembly Connections

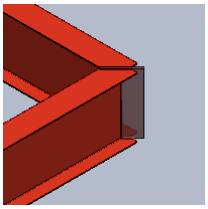
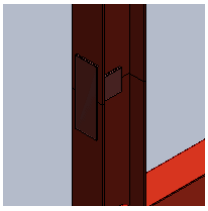
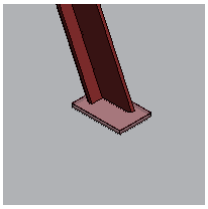
Member assembly connections are similar to frame connections, but define the necessary trimming between member parts and the generation of parts such as base plates, gusset plates, and clip angles. Assembly connections control member features including cutbacks, copes, notches, bolt holes, and slots. Whether or not features are placed depends on the member assembly connection type and the geometry of the connection between the members.

There are several basic assembly connections delivered with the software. You can create your own assembly connections by editing the **StructAssemblyConnections.xls** workbook, and then bulk loading the workbook. For more information on creating your own assembly connections, see the Structure Reference Data Guide.

NOTE For very complex nodal connections, instead of writing a custom assembly connection, you might want to try *Trim Members* (on page 201) in the Structure task.

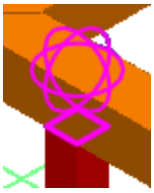
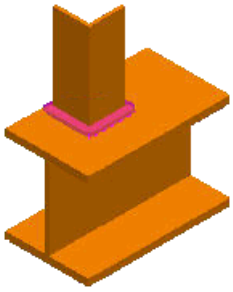
The following general assembly connections are delivered with the software:

	The base plate assembly connection places a plate at the end of an unsupported member. This assembly connection requires an unsupported frame connection on one member. For example, use this assembly connection to place a base plate at the bottom of a column.
	The corner gusset plate assembly connection connects a vertical brace to a beam and column intersection using a gusset plate. This assembly connection requires a frame connection with three members, such as vertical corner brace.
	The fitted assembly connection connects two members. This assembly connection requires a frame connection with two members, such as axis, seated, or flush. Examples of this connection include a beam framing into a column or a beam framing into another beam.
	The gusset plate assembly connection connects a vertical or horizontal brace to a beam or a vertical brace to a column using a gusset plate. This assembly connection requires a frame connection with two members, such as axis.

	The miter assembly connection connects two members that meet at an angle but are co-planar. This assembly connection requires a frame connection with two members, such as axis. In addition, the members must be end connected.
	The splice assembly connection connects two members that are collinear and end connected. This assembly connection requires a frame connection with two members, such as axis.
	The general surface assembly connection connects a member end with a nonmember surface such as a slab, a deck, or a hull plate. The member is cut to surface and a base plate is placed on the member end.

End Cut Assembly Connections

The following end cut assembly connections are delivered with the software:

	The end cut along assembly connection cuts the end of one member to the sides or bottom of another member along the second member's length.
	The end cut bearing plate assembly connection connects the end of a member along the axis of another member. This is typically used to attach the end of a member to the flange of the supporting member. The bearing plate connection works at any angle and provides circular, rectangular, and triangular bearing plates depending on the supported member's shape.
	The end cut miter assembly connection connects two members that meet at an angle but are co-planar. This assembly connection required a frame connection with two members, such as axis. In addition, the members must be end connected.

	The end cut split assembly connection connects two members that are collinear and end connected. This assembly connection requires a frame connection with two members, such as axis.
	The end cut split long box assembly connection is used when the supported and supporting members are perpendicular to each other. The supporting member is cut square to the supported member while an item from the Axis category is placed on the supported member.
	The end cut split seam angle assembly connection is used when the supported and supporting members are: ▪ member axes are collinear to each other ▪ have identical cross-section type and size ▪ have identical cardinal point and rotation values
	The end cut split short box assembly connection is used when the supported and supporting members are perpendicular to each other. The supported member is cut square to the supporting member while an item from the Axis category is placed on the supporting member.

End Cut Assembly Naming Conventions

Each end cut assembly connection name consists of three parts:
[EndCutCase]_[WebCutType]_[FlangeCutType]

where:

- [EndCutCase]
  - **Axis** - Supported member is bounded by the axis curve of the supporting member
  - **LongBox** - End-to-end perpendicular supported member is extended, and an axis end cut is applied to the supporting member
  - **ShortBox** - End-to-end perpendicular supporting member is extended, and an axis end cut is applied to the supported member
  - **Miter** - End-to-end non-collinear
  - **Split** - End-to-end collinear
  - **SeamAngle** - End-to-end collinear where one end is extended
- [WebCutType]

  The WebCutType is used to type of web cut that is to be applied at the top of the web and at the bottom of the web. The following combinations are available (W indicates a straight cut, C indicates a Cope cut, and S indicates a Snipe cut):

  - W#W#
  - W#C#
  - W#S#
  - C#C#

- C#S#
- C#W#
- S#S#
- S#C#
- S#W#

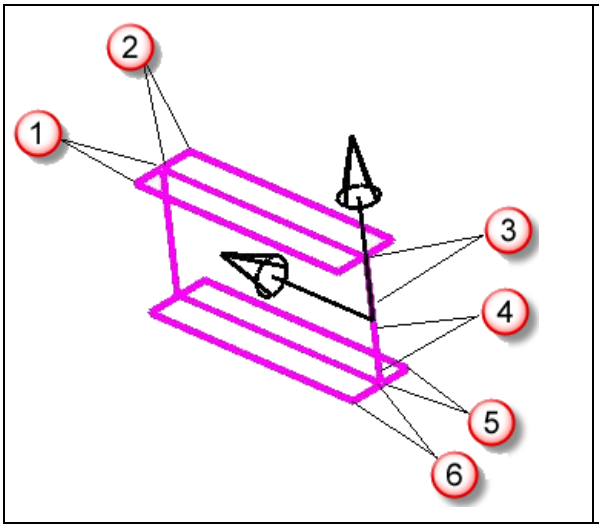
- [FlangeCutType]

  The FlangeCutType is used to type of flange cut that is to be applied at the left flange section and at the right flange section. The following combinations are available (W indicates a straight cut, C indicates a cope cut, and S indicates a snipe cut):

  - W#W#
  - W#S#
  - S#S#
  - S#W#

You cannot apply different flange cuts when the supported member has both a top flange and a bottom flange. In these cases, the same flange cut type is applied to both the top and bottom flange.

You cannot apply different web cuts or flange cuts in the end-to-end cases. The end-to-end cases place a web cut and flange cut on both the supported member and the supporting member. For these cases, the same web cut and flange cut are applied to both the supported member and the supporting member.



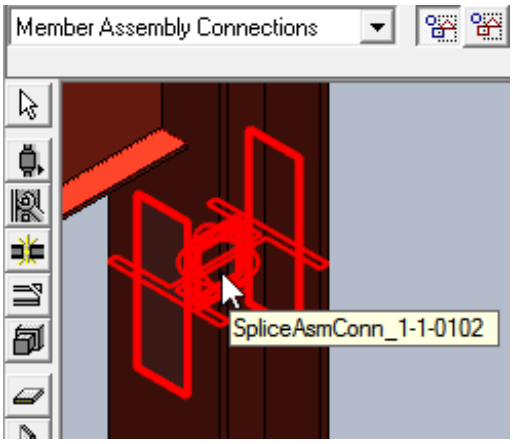
1 - Top left flange
2 - Top right flange
3 - Web top
4 - Web bottom
5 - Bottom right flange
6 - Bottom left flange

See *Members* (on page 22) in the Structure task for important related information.

Locating Assembly Connections

Assembly connections do not display in the model. However, if you set the **Locate Filter** to **Member Assembly Connections** in the Structure task, you can locate and select assembly connections for review and editing. Assembly connections are located at the ends of member

parts and appear as circles when you move the cursor over them. Any assembly components, such as gusset plates, associated with the assembly connection also highlight.



When you select an assembly connection, the software displays the assembly connection type in the ribbon. Select the **Edit > Properties** command to edit the assembly connection properties. You cannot measure distances using the assembly connection graphic as an end point when using the **Tools > Measure** command.

Slab Assembly Connections

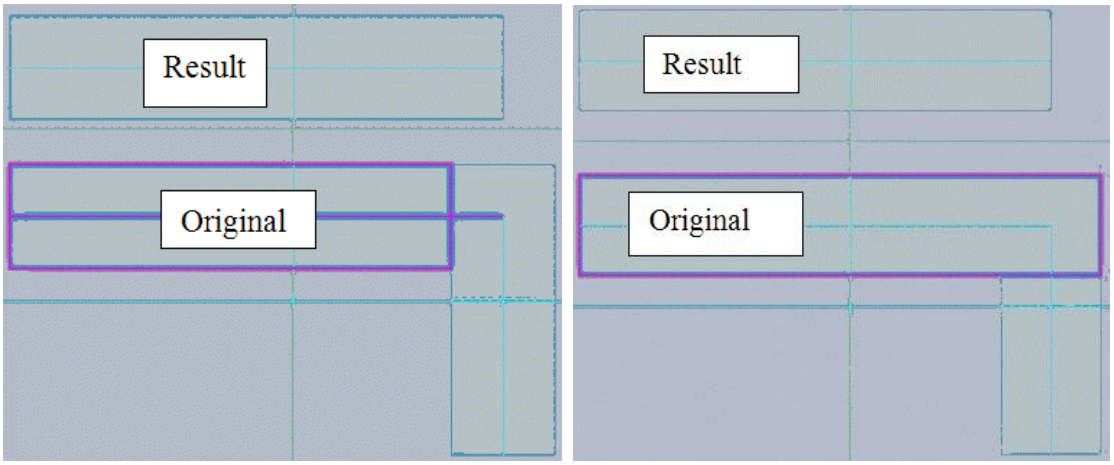
Slab assembly connections are similar to member assembly connections, but define the trimming and edges of slabs. Slab assembly connections are placed automatically when the slab is created.

The slab assembly connections are:

- The Slab by Member Boundary assembly connection is placed by the software when you select a member as a slab boundary.
- The Slab Free Edge Assembly connection is placed by the software when you select a grid line as a slab boundary.

If a slab's bounding object (for example a member, a wall, or another slab) is deleted, then that deleted object is replaced as a boundary by the slab assembly connection which is generated from that deleted object. This behavior keeps the slab from going to the **To Do List** as missing a boundary.

In addition, if you copy, move, or mirror a wall (including Model Data Reuse operations) and an object which was connected to the wall is not in the copy set and the **Delete optional** option is selected, the software changes the wall assembly connection for the missing object to be a Wall Free Edge Assembly connection. This prevents the wall from going to the **To Do List** with a "Wall is missing a boundary Select a valid boundary" message. Be aware that copying or moving a wall with the **Delete optional** option selected can cause the geometry of the wall to extend or shorten.



IMPORTANT This geometry change impacts the use of the **Validate Model Data Reuse Results** command because the geometry of the wall is different between the original wall and the new wall.

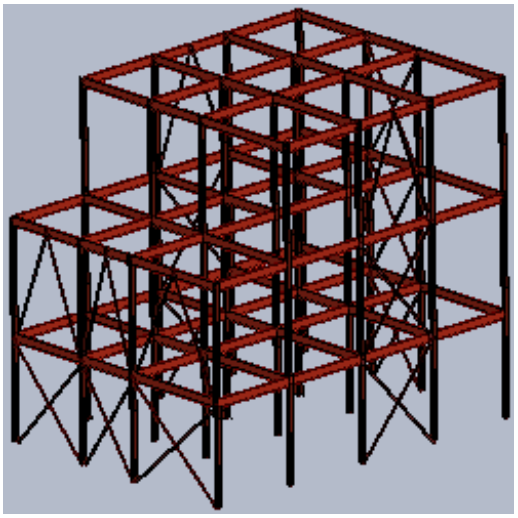
Locating Slab Assembly Connections

Assembly connections do not display in the model. However, if you set the **Locate Filter** to **Slab Assembly Connections**, you can locate and select assembly connections for review and editing. Assembly connections are located at the edges of slabs.

When you select an assembly connection, the software displays the assembly connection type in the ribbon. Select the **Edit > Properties** command to edit the assembly connection properties.

SECTION 3

Place Linear Member Systems



Places a linear member in the model. You can place beams, columns, braces, truss elements, or cable member types using this command. You can place standard members (rolled shapes) or designed members (built-up shapes). For additional information, see *Members* (on page 22). Use this command when you want to place members by specifying the exact start and end points.

NOTES

- You can define custom member types by editing the **Structural Member Type** select list in the Catalog task.
- When a linear member system is connected directly to another member, a frame connection is created. If detailed trimming is needed in a plant database, use the **Trim Member** or **Place Assembly Connection**. If detailed trimming of the members is needed in a marine database, use **Place Member Generic Assembly Connection** in the Molded Forms or Structural Detailing task.
- When a linear member system is connected to a plate system or edge reinforcement system, the member is trimmed to the plate and a generic assembly connection is created. In marine mode, the bounding ports can be modified with **Place Member Generic Assembly Connection** in the Molded Forms or Structural Detailing task. For more information, see Place Member Generic Assembly Connection.

Place Linear Member System Ribbon

Specifies the properties for the member that you are placing. When editing a member part, this ribbon changes. For more information on the properties that are available when you are modifying a member part, see *Modify Linear Member Part Ribbon*.

Member Properties

Activates the *Member System Prismatic Properties Dialog Box* (on page 61). You can use this dialog box to specify additional member properties that you cannot set on the ribbon, such as material, material grade, and end releases. Because changes made on the ribbon, such as to the **Type Category**, can affect or reset properties in the dialog box, we recommend that you set properties in this dialog box after you have made your selections from the ribbon.

Start

Specify the start location of the member. After placing the first member, click **Start** to select the discrete placement method. For more information about discrete placement, see *Members* (on page 22).

End

Specify the end location of the member. After placing the first member, click **End** to select the contiguous placement method. For more information about contiguous placement, see *Members* (on page 22).

Enter AMP

Activates the *Advanced Member Positioning Ribbon*, which you can use when you want to place a member at the intersection of three objects and maintain the relationship to those objects. For example, a column that needs to stay over two walls, or bulkheads, on a lower level. You must select the **Align-Default** or **Align-Lapped** frame connections from the **Connection** box to use the advanced member positioning options.

Finish

Click to place the member in the model. **Finish** is active only when **Finish Mode** is selected.

Finish Mode

Specify whether or not **Finish** must be selected to place a member in the model. If the **Finish Mode** is selected, the software places the member in tentative mode after you identify the second end point. This tentative mode allows you to modify placement settings such as the offset, cardinal point, or frame connection properties before you commit the member to the model. If the **Finish Mode** is not selected, then the software automatically places the member in the model after you identify the second end point.

Connection

Select the frame connection type to use for the member that you are placing. If you select **By Rule**, the software determines the frame connection to use based on the geometry between the member that you are placing and existing members in the model. If you select **More**, all available frame connections display from which you can select the frame connection to use. For more information about frame connections, see *Frame Connections* (on page 24). This option is not available if you are editing an existing member. You can change the frame connection later by editing the frame connection directly.

Connection Properties

Activates the **Connection Properties** dialog box, which is used to specify properties for the active frame connection. The properties that appear in this dialog box are described under the Connection Properties section.

System

Select the system to which the member belongs. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see *Select System Dialog Box* in *Place Linear Member Systems* (on page 35).

Type Category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

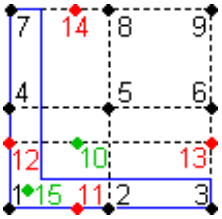
Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Section Name

Defines the cross-section for the member. If you know the section name, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See Structure Reference Data Guide for more information about reference data.

Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

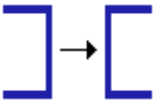


Angle

Defines the angle, in degrees or radians, by which the cross-section is rotated about the member axis. The zero-degree position is either the Z-axis or the X-axis of the active coordinate system depending on the member orientation. The active coordinate system is set using the **Coordinate System** control in **PinPoint**  . The **Angle** property is not used when placing members using the **Seated**, **Flush**, or **Centerline** frame connections.

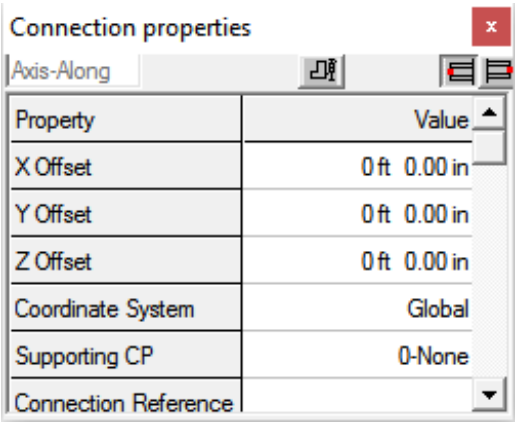
Reflect

Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.



Connection Properties

The **Connection Properties** appear only when you have selected the **Connection Properties** option. Connection properties change depending on the frame connection specified in the **Connection** option. To see the frame connection properties for the start of the member, select . To see the frame connection properties for the end of the member, select . Click to see a preview of the frame connection. The frame connection type appears in the upper left corner of the dialog box.



TIP The supported member is the member that you are placing. The supporting member is the existing member in the model to which you are connecting.

Seated, Flush, and Centerline Frame Connection Properties

Side

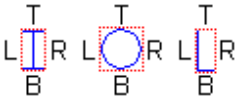
Select the side of the supporting member on which you want to place the supported member.

Offset

Specify the distance to place the supported member from the supporting member. For seated and flush frame connections, the offset is between the side of the supporting member that you specified with the **Side** option and the supported member's side that you specify with the **Edge** option. For centerline frame connections, the offset is between centerline of the supporting member and the supported member's side that you specify with the **Edge** option.

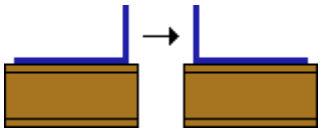
Edge

Specifies the side of the supported member's cross-section that is mated to the supporting member. You can specify **Top**, **Right**, **Bottom**, or **Left**. Edges of typical section shapes are shown in the figure.



Reflect

Reflects or mirrors the cross-section of the supported member about a plane perpendicular to the supporting member side. For example, when you place a supported member with an angle cross-section using the left edge option and you want the angle facing the other direction.



Axis Frame Connection Properties

X Offset

Specifies an offset to apply in the x-direction after the two cardinal points are aligned.

Y Offset

Specifies an offset to apply in the y-direction after the two cardinal points are aligned.

Z Offset

Specifies an offset to apply in the z-direction after the two cardinal points are aligned.

Coordinate System

Specifies the coordinate system to use for the offset values. Local is the local coordinate system of the supporting member.

Supporting CP

Specifies to which cardinal point on the supporting member system to align the supported member system's cardinal point. You can specify any cardinal point number, or select 0 to use the cardinal point with which the supporting member was placed.

Surface Frame Connection Properties

X Offset

Specifies an offset to apply in the x-direction.

Y Offset

Specifies an offset to apply in the y-direction.

Z Offset

Specifies an offset to apply in the z-direction.

Coordinate System

Specifies the coordinate system to use for the offset values.

Vertical Corner Brace Frame Connection Properties

X Offset

Specifies the offset to apply in the x-direction.

Y Offset

Specifies the offset to apply in the y-direction.

Z Offset

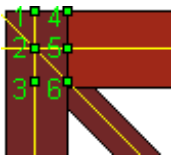
Specifies the offset to apply in the z-direction.

Coordinate System

Select whether the offset values are defined relative to the global coordinate system or the member's local coordinate system.

Work Point

Specifies the work point location. There are six work point locations that you can select.



1 - Primary Center - Secondary Far Side
2 - Primary Center - Secondary Center
3 - Primary Center - Secondary Near Side
4 - Primary Near Side - Secondary Far Side
5 - Primary Near Side - Secondary Center
6 - Primary Near Side - Secondary Near Side

Modify Linear Member Part Ribbon

Displays the member part properties that you are editing when you select one or more linear member parts.

Member Properties

Activates the *Member Part Prismatic Properties Dialog Box* (on page 67). You can use this dialog box to specify additional member part properties, such as material, material grade, and end releases, which you cannot set on the ribbon.

Start

Specify the start location of the member. After placing the first member, click **Start** to select the discrete placement method. For more information about discrete placement, see *Members* (on page 22).

End

Specify the end location of the member. After placing the first member, click **End** to select the contiguous placement method. For more information about contiguous placement, see *Members* (on page 22).

Convert

Translates a member part to have a single stand-alone member system. This option is only available when you select a member part that belongs to a member system that has been split.

When you place a member system, that member system has a single member part associated with it (for more information, see *Members* (on page 22)). Using the *Place Splits* (on page 191), you can split that single member part into multiple member parts that are each still associated with the original single member system.

If you want to delete or modify only one of the member parts, you must delete the member system and all the other member parts. Deleting a member part deletes its parent member system which causes the sibling member parts to be deleted. However, using the **Convert** option you can cause a member part to have its own, new parent member system. You can then delete or otherwise modify that member part without affecting the other member parts of the original member system.

The software automatically translates the relevant split connections into frame connections. Permission groups of the original member system are used for the new member system.

System

Select the system to which the member belongs. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see *Select System Dialog Box* in *Place Linear Member Systems* (on page 35).

Type Category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

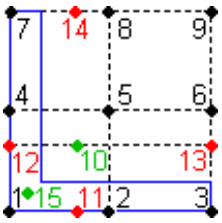
Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Section Name

Defines the cross-section for the member. If you know the section name, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See Structure Reference Data Guide for more information about reference data.

Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

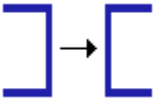


Angle

Defines the angle, in degrees or radians, by which the cross-section is rotated about the member axis. The zero-degree position is either the Z-axis or the X-axis of the active coordinate system depending on the member orientation. The active coordinate system is set using the **Coordinate System** control in **PinPoint**  . The **Angle** property is not used when placing members using the **Seated**, **Flush**, or **Centerline** frame connections.

Reflect

Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.



Advanced Member Positioning Ribbon

Specifies the properties for the member that you are placing using the advanced member positioning option. When editing a member part, this ribbon changes. For more information on the properties that are available when you are modifying a member part, see *Modify Linear Member Part Ribbon* in *Place Linear Member Systems* (on page 35).

 **Member Properties**

Activates the *Member System Prismatic Properties Dialog Box* (on page 61). You can use this dialog box to specify additional member properties that you cannot set on the ribbon, such as material, material grade, and end releases. Because changes made on the ribbon, such as to the **Type Category**, can affect or reset properties in the dialog box, we recommend that you set properties in this dialog box after you have made your selections from the ribbon.

 **Start**

Specify the start location of the member. After placing the first member, click **Start** to select the discrete placement method. For more information about discrete placement, see *Members* (on page 22).

 **End**

Specify the end location of the member. After placing the first member, click **End** to select the contiguous placement method. For more information about contiguous placement, see *Members* (on page 22).

 Exit AMP

Exits the advanced member positioning ribbon, and returns you to the standard member controls.

 **Connection Properties**

Activates the **Connection Properties** dialog box, which is used to specify properties for the active frame connection. The properties that appear in this dialog box are described below in the *Connection Properties* section.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.
- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available. For more information, see *[Product Folder]*\Programming\Help\GeometricConstructions.chm, available when Programming Resources is installed.

1

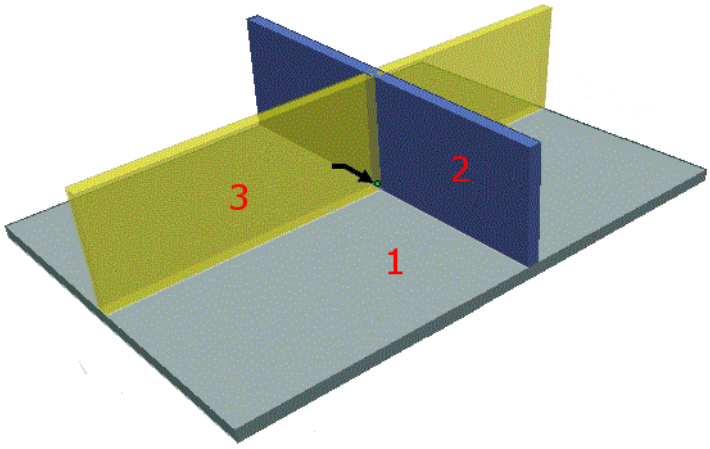
Specifies the bounding object.

2

Specifies the align object.

3

Specifies the intersect object.

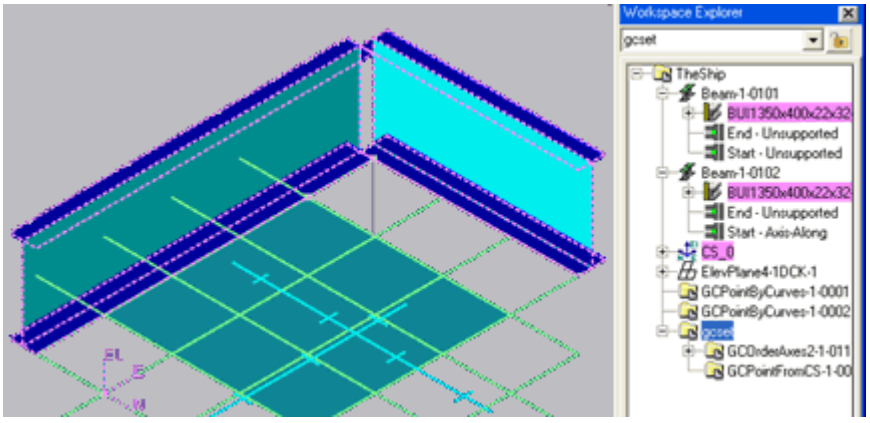


Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.
- 1 - Yellow background when an input is selected.
- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:



When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.
- 1 and 0.10 m - Yellow background when you change a value.
- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

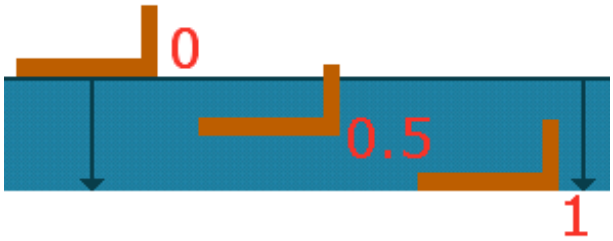
IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Bound Offset

Type an offset distance for the point from the bound surface. This offset is in addition to the ratio you defined.

Bound Ratio

Type the ratio of the plate's thickness to the point position. Type **0** to place the point on the molded-form surface. Type **1** to place the point on the anti-molded-form surface. Entering a ratio of 0.5 centers the point between the two surfaces.



Bound Orient

Specifies how to interpret the bounding object for advanced member positioning. Select **Primary** to evaluate along the primary axis. Select **Secondary** to evaluate along the secondary axis. See **Orient Options** below.

Align Offset

Type an offset distance for the point from the align surface. This offset is in addition to the ratio you defined.

Align Ratio

Type the ratio of the plate's thickness to the point position. Type **0** to place the point on the molded-form surface. Type **1** to place the point on the anti-molded-form surface. Entering a ratio of 0.5 centers the point between the two surfaces.

Align Orient

Specifies how to interpret the align object for advanced member positioning. Select **Primary** to evaluate along the primary axis. Select **Secondary** to evaluate along the secondary axis. See **Orient Options** below.

Intersect Offset

Type an offset distance for the point from the intersect surface. This offset is in addition to the ratio you defined.

Intersect Ratio

Type the ratio of the plate's thickness to the point position. Type **0** to place the point on the molded-form surface. Type **1** to place the point on the anti-molded-form surface. Typing a ratio of **0.5** centers the point between the two surfaces.

Intersect Orient

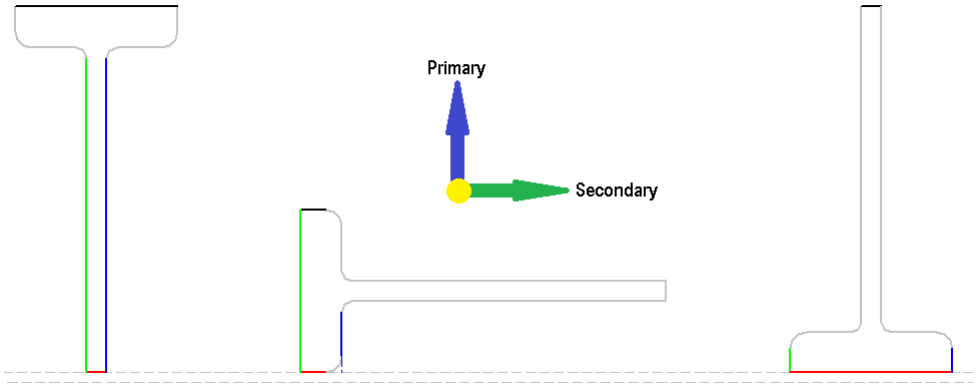
Specifies how to interpret the intersect object for advanced member positioning. Select **Primary** to evaluate along the primary axis. Select **Secondary** to evaluate along the secondary axis. See **Orient Options** below.

Orient Options

Specifies if the profile is interpreted along its primary orientation or its secondary orientation. If along the primary orientation, then 0.0 is the red surface and 1.0 is the black surface. If along the secondary orientation, then 0.0 is the green surface and 1.0 is the blue surface.

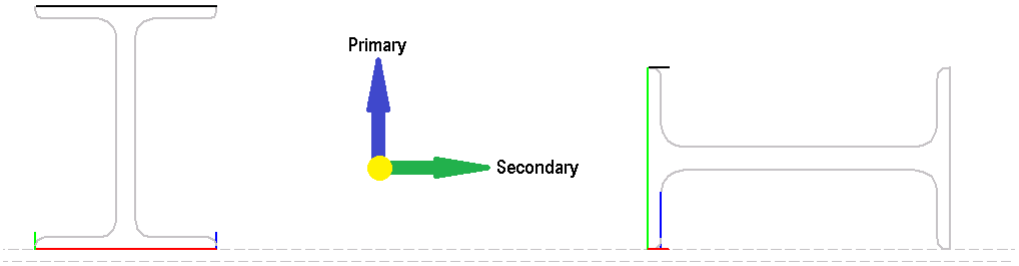
Tee

Includes W, S, and M types and fabricated Ts



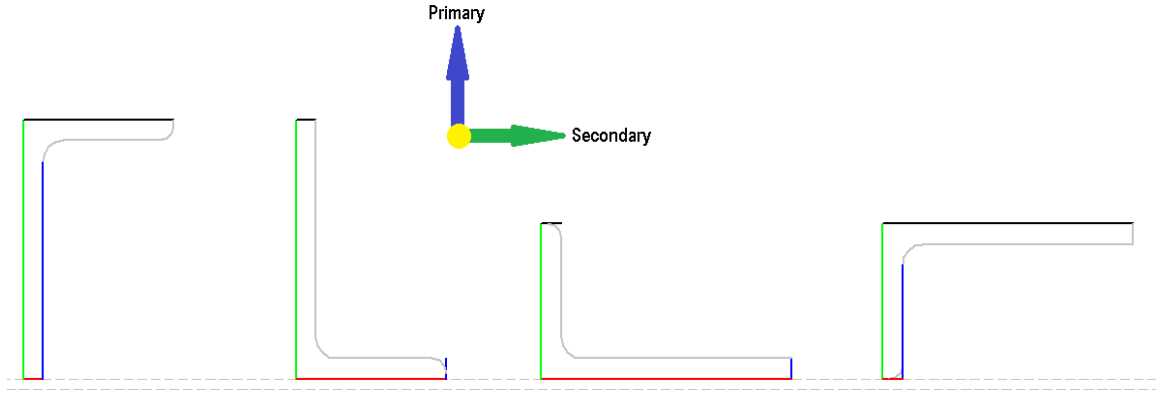
I-beams

Includes W and S types.

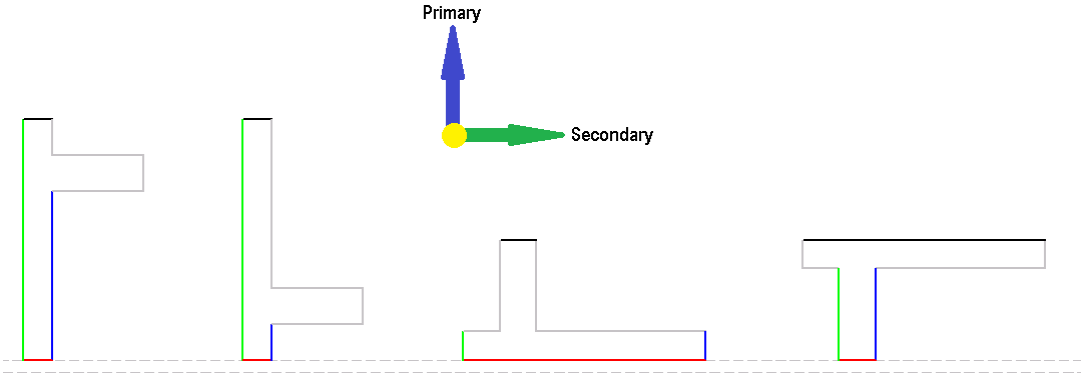


Angle

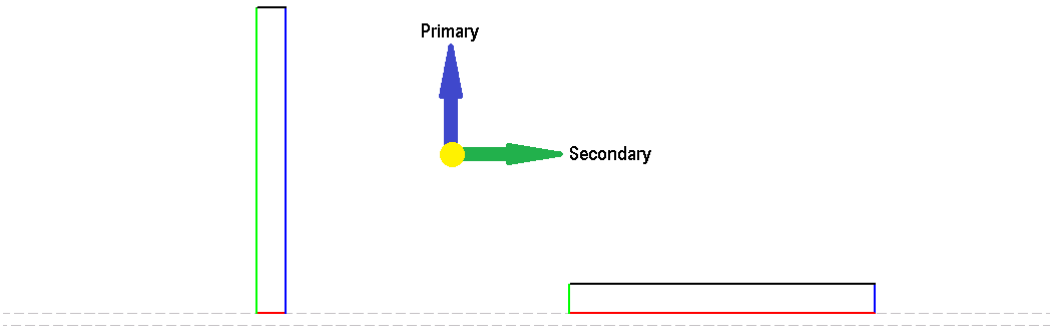
Includes equal and unequal leg angles.



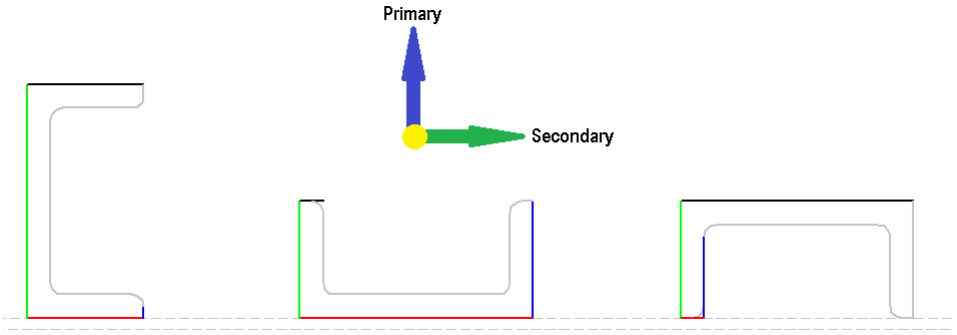
Fabricated Angle



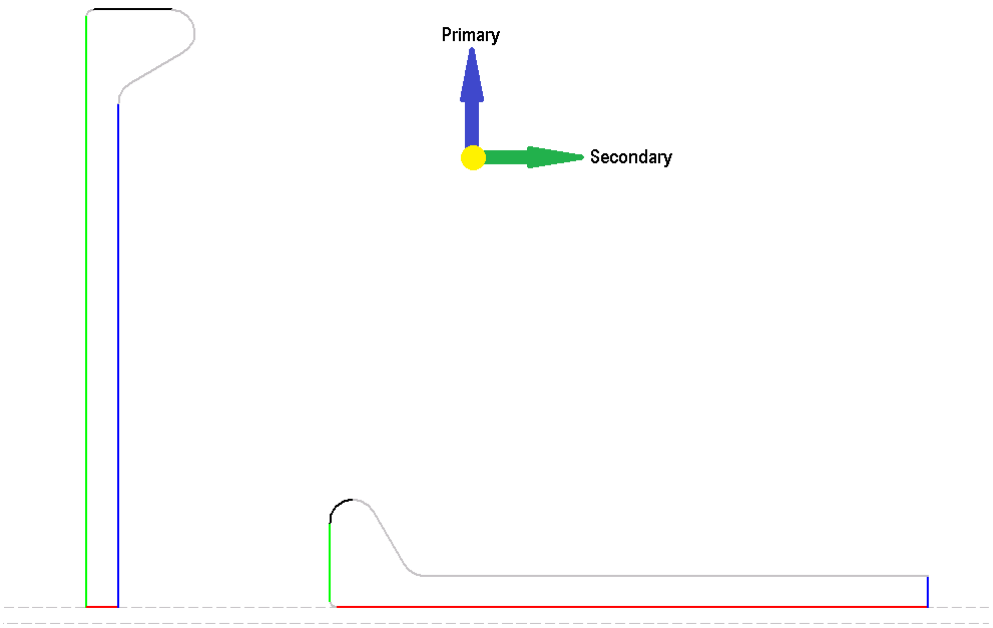
Flat Bar



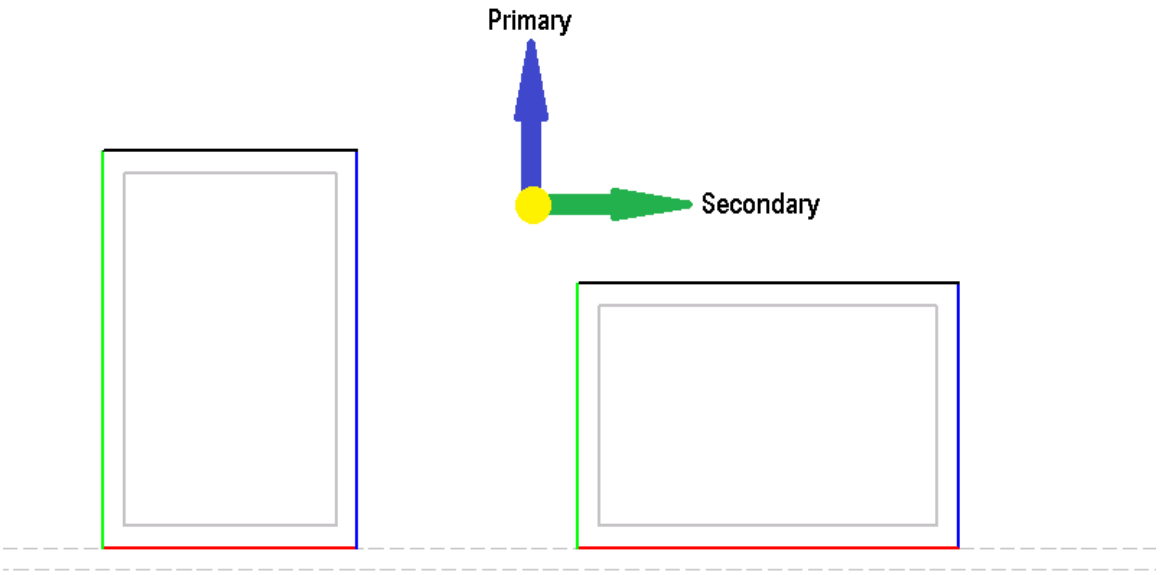
Channel



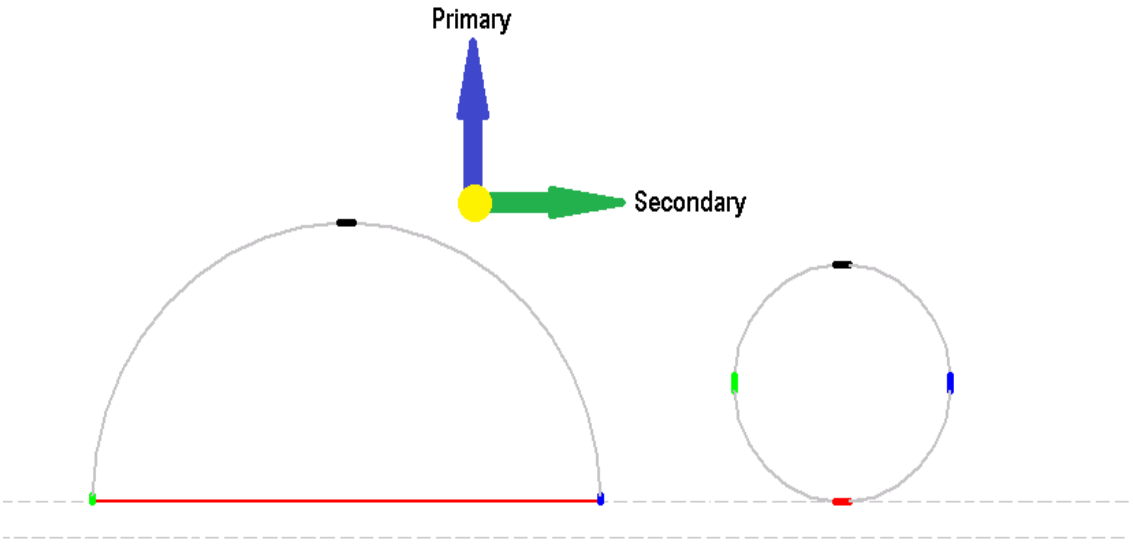
Bulb



Rectangular Tube

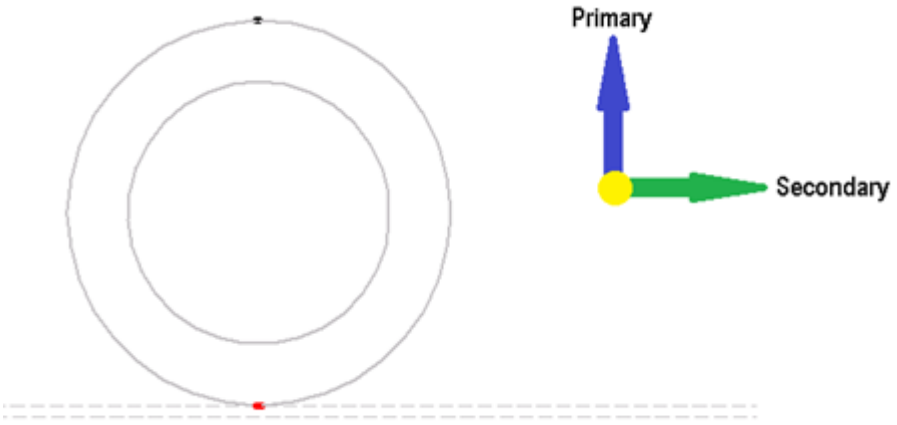


Half Round and Round Bar



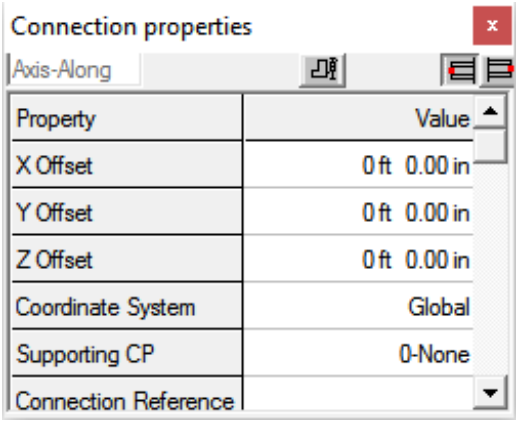
Round Tube (Pipe)

NOTE Round Tube has no faces defined along the secondary orientation.



Connection Properties

The **Connection Properties** appear only when you have selected the **Connection Properties** option. Connection properties change depending on the frame connection specified in the **Connection** option. To see the frame connection properties for the start of the member, select . To see the frame connection properties for the end of the member, select . Click to see a preview of the frame connection. The frame connection type appears in the upper left corner of the dialog box.



TIP The supported member is the member that you are placing. The supporting member is the existing member in the model to which you are connecting.

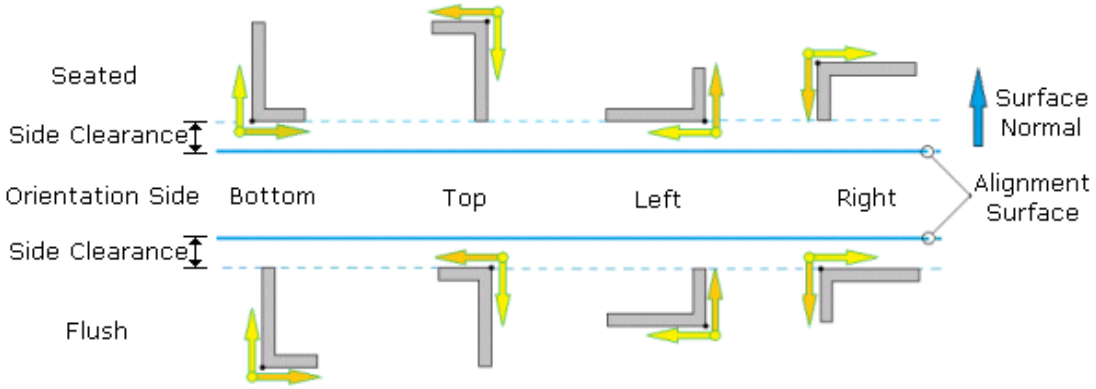
Align-Default/Lapped Frame Connection Properties

Side Justification

Select the side of the plane on which to place the member.

- **None** - Aligns the member axis so that it is placed on the surface.
- **Seated** - Sets the member on top of the surface. The selected **Orientation Side** is nearest the surface.

- **Center** - The center of the cross-section is placed on the surface. The center is calculated by the top-bottom, left-right bounds of the cross-section shape.
- **Flush** - Uses the alignment surface's top and bottom extent to position the supported member. The supported member typically lies within the body of the alignment plane with one edge of the member flush with the alignment surface but can be offset.

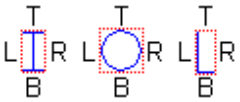


Orientation Control

If **On**, the member angle is controlled by the **Orientation Angle** value defined for the Align frame connection. If **Off**, you can set the angle of the member independently of the align surface by using the **Rotation** property for the member part.

Orientation Side

Specifies the side of the supported member's cross-section that is mated to the alignment surface. You can specify **Top**, **Right**, **Bottom**, or **Left**. Edges of typical section shapes are shown in the figure.

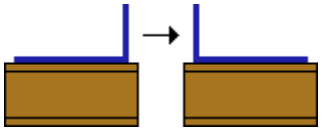


Orientation Angle

When **Orientation Control** is **On**, this angle is added to the selected **Orientation Side** to rotate the member. If the Align frame connection is used on both ends of the member and you define a different angle value for each end, the angle defined at the start end of the member is used.

Reflect

Reflects or mirrors the cross-section of the supported member about a plane perpendicular to the supporting member side. An example of when to use this option is when you place a supported member with an angle cross-section using the left edge option and you want the angle facing the other direction.



Extend Distance

Specifies the offset distance from the bounding surface along the member's axis.

Side Clearance

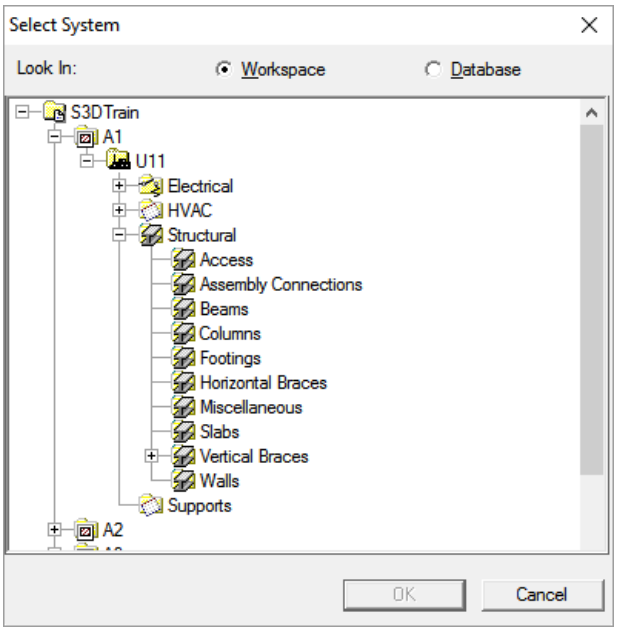
Specify the offset distance perpendicular to the align surface to place the member axis.

Lateral Distance

Specifies the offset distance in the plane of the align surface to place the member axis.

Select System Dialog Box

This dialog box displays when you select the **More...** option in the **System** list. Use this dialog box to select the system that you want. You can create new systems in the Systems and Specifications task.



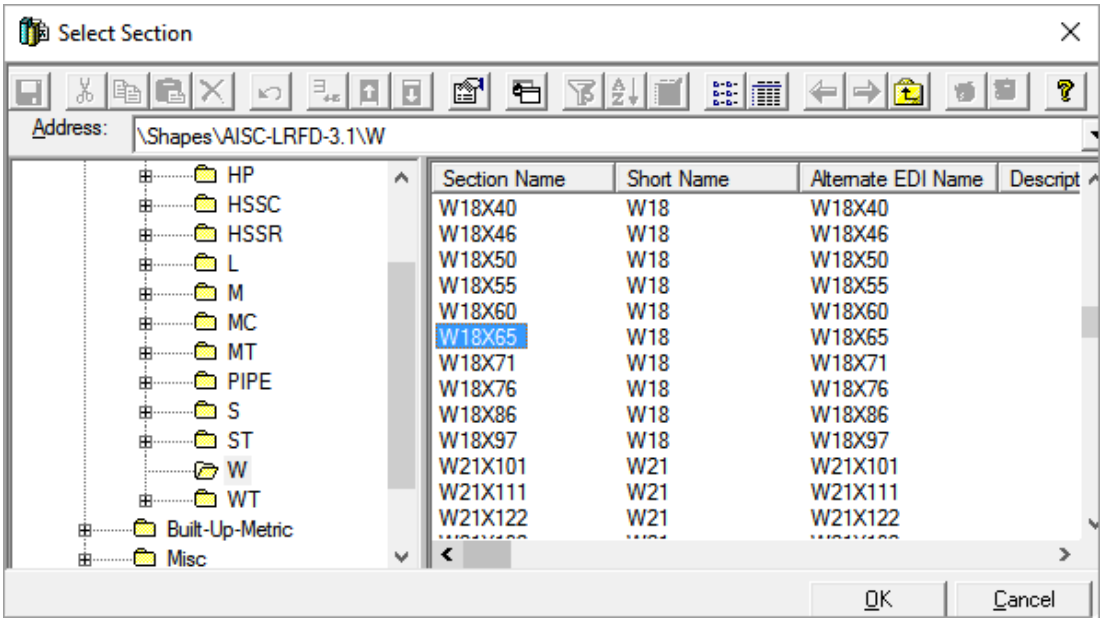
Look in

Specify where you want to look for the system. Select **Workspace** to look for the system in your defined workspace only. Select **Database** to look for the system in the entire Model database.

Select Section Dialog Box

Allows selection of the type of section to be placed. This dialog box appears when you click the **More** option on the **Section Name** list. By browsing through the hierarchy, you can find any

section in the Catalog database. After you select a section, the software returns you to the model, where you can finalize placement.



Properties

Displays the properties of the selected section. Because you cannot modify any properties until the section is placed, all properties on the dialog box are read-only.

Preview

Displays a picture of the selected section. The image file must be assigned to the section in the reference data.

Filter

Allows you to filter catalog data to help find the subset of data that you want to work with, similar to Microsoft Excel.

Sort

Sorts the catalog data by column to help you find like items.

Customize Current View

Defines with columns in the data you want to see.

List View

Sets the dialog box to display sections in a list view.

Grid View

Sets the dialog box to display sections in a spreadsheet-style grid view.

Back

Returns you to the previously selected section type or node. Use this command to navigate through the hierarchy to the specific type that you need.

Forward

Sends you to the last selected section type or node that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific type that you need.

Up One Level

Brings up the next highest level of the catalog hierarchy. Use this command to navigate through the hierarchy to the specific type that you need.

Address

Specifies your exact location within the displayed hierarchy.

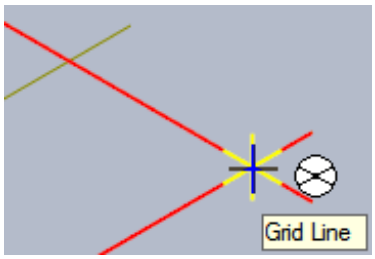
What do you want to do?

- *Place members using discrete placement* (on page 54)
- *Place members using contiguous placement* (on page 55)
- *Place a member using finish mode* (on page 56)
- *Edit member system properties* (on page 57)
- *Edit member part properties* (on page 57)
- *Edit a frame connection* (on page 57)
- *Delete a member system* (on page 57)
- *Convert a member part* (on page 58)
- *Modify the cardinal point of a member* (on page 58)
- *Modify the end releases of a member* (on page 58)
- *Modify the angle of a member* (on page 59)
- *Modify the cross-section of a member* (on page 59)
- *Modify the material of a member* (on page 59)
- *Modify the material grade of a member* (on page 59)
- *Modify the type of member* (on page 60)
- *Move a member* (on page 60)
- *Move one end of a member* (on page 60)
- Move one end of multiple members

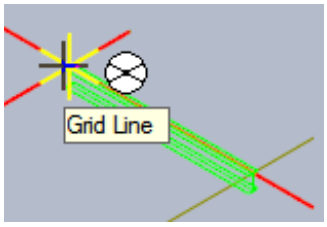
Place members using discrete placement

1. Click **Place Linear Member System** on the vertical toolbar.
2. In the **Connection** box, select a frame connection type.

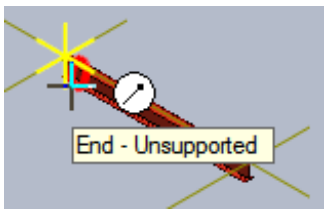
   TIP If you are unsure of which frame connection type to use, review *Frame Connections* (on page 24). You can also select the **By Rule** option to allow the software to select automatically a frame connection type.
3. In the **Type category** box, select the member type category to place.
4. In the **Type** box, select the member type to place.
5. Specify the start location, or first point, of the member.

   TIP You can use the frame connection of another member as the start or end location of the member that you are placing.
6. Specify the end location, or second point, of the member.

7. Click **Start** to specify the start location of the next member.
8. Specify the start location of the next member.

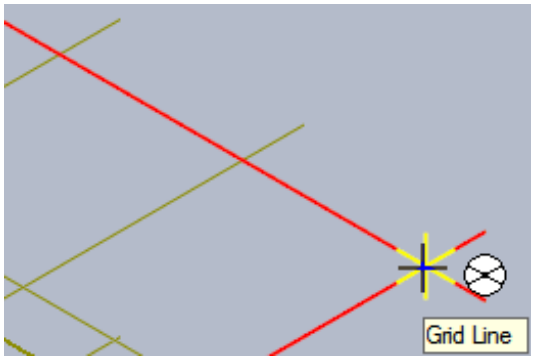
NOTES

- To find the intersection of grid lines easily, click **Tools > Options** to verify that the SmartSketch **Intersection** option is selected.
- You can set the beta angle relative to your rotated coordinate system by activating **PinPoint** and selecting the rotated coordinated system as the active coordinate system in the **Coordinate System** box.

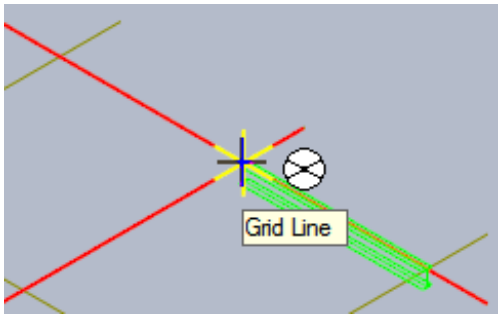
Place members using contiguous placement

1. Click **Place Linear Member System** on the vertical toolbar.
2. In the **Connection** box, select a frame connection type.

   TIP If you are unsure of which frame connection type to use, review *Frame Connections* (on page 24). You can also select the **By Rule** option to allow the software automatically to select a frame connection type.
3. In the **Type category** box, select the member type category to place.
4. In the **Type** box, select the member type to place.
5. Specify the start location, or first point, of the member.

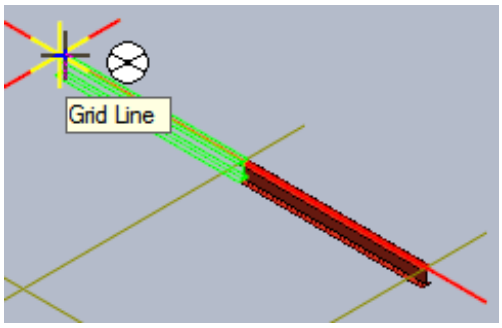
   

   TIP You can use the frame connection of another member as the start or end location of the member that you are placing.
6. Specify the end location, or second point, of the member.

7. Click **End** to activate the contiguous placement mode.

8. Specify the end location of the next member.



NOTE To find the intersection of grid lines easily, click **Tools > Options** to verify that the SmartSketch **Intersection** option is selected.

Place a member using finish mode

1. Click **Place Linear Member System** on the vertical toolbar.
2. In the **Connection** box, select a frame connection type.

   TIP If you are unsure of which frame connection type to use, review *Frame Connections* (on page 24). You can also select the **By Rule** option to allow the software to select automatically a frame connection type.
3. Click **Finish Mode** to activate the **Finish** button.
4. Click **Connection Properties** to activate the **Connection Properties** dialog box.
5. In the **Type category** box, select the member type category to place.
6. In the **Type** box, select the member type to place.
7. Specify the start location, or first point, of the member.

   TIP You can use the frame connection of another member as the start or end location of the member that you are placing.
8. Specify the end location, or second point, of the member.
9. Edit the start and end frame connection properties in the **Connection Properties** dialog box.
10. Edit the member properties using the **Member Properties** dialog box.
11. Click **Finish**.

NOTES

- To find the intersection of grid lines easily, click **Tools > Options** to verify that the SmartSketch **Intersection** option is selected.
- You can set the beta angle relative to your rotated coordinate system by activating **PinPoint** and selecting the rotated coordinated system as the active coordinate system in the **Coordinate System** box.

Edit member system properties

1. Click **Select** on the vertical toolbar.
2. Select **Members System** in the **Locate Filter**.
3. Select the member system to edit.
4. Click **Edit > Properties**.
5. Edit the properties as needed. For more information about properties, see *Member System Prismatic Properties Dialog Box* (on page 61).

Edit member part properties

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select the member part to edit.
4. Click **Edit > Properties**.
5. Edit the properties as needed. For more information about properties, see *Member Part Prismatic Properties Dialog Box* (on page 67).

Edit a frame connection

1. Click **Select** on the vertical toolbar.
2. Select **Frame Connections** in the **Locate Filter**.
3. Select the frame connection to edit.
4. Edit the frame connection as needed. For more information about frame connection properties, see *Frame Connection Properties Dialog Box* (on page 99).

Delete a member system

1. Click **Select** on the vertical toolbar.
2. Select **Members System** in the **Locate Filter**.
3. Select the members to delete.
4. Click **Delete**.

NOTES

- All loads and boundary conditions placed in the Structural Analysis task on the deleted member system are also deleted. This could affect any Analytical Models that have been exported.
- All footings associated with the member system are also deleted.

- Ladders or stairs using the deleted member system as the defined top edge are sent to the **To Do List**.

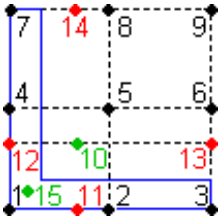
Convert a member part

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member part to have its own member system.
4. Click **Convert** on the ribbon.

Modify the cardinal point of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select the member part to modify.
4. Using the **Cardinal Point** control on the ribbon bar, select a new cardinal point for the member part.

NOTE There are 15 cardinal positions available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13.



Modify the end releases of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Click **Member Properties** on the ribbon.
5. Select the **Member Part** tab.
6. In the **Category** box, select **End Releases**.
7. Modify the member part end releases as required.

Modify the angle of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Using the **Angle** control on the ribbon bar, edit the member angle.

   TIP You can set the beta angle relative to your rotated coordinate system by activating **PinPoint** and selecting the rotated coordinated system as the active coordinate system in the **Coordinate System** box.

Modify the cross-section of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Using the **Section name** box on the ribbon bar, type the new cross-section name for the member, or select **More** to select the cross-section from the Catalog.

Modify the material of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Click **Edit > Properties**.
5. Select the **Cross Section** tab.
6. Select a new value for the **Material** property.

Modify the material grade of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Click **Edit > Properties**.
5. Select the **Cross Section** tab.
6. Select a new value for the **Grade** property.

Modify the type of member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Using the **Type Category** and **Type** controls on the ribbon bar, change the type of the member part.

NOTE If the member type category or type that you want to use is not available, you can add it by editing the **Structural Member Type** sheet in the **AllCodeLists.xls** workbook and bulkloading it into the Catalog database. For more information on editing the workbook and bulk loading, see the *Smart 3D Reference Data Guide*.

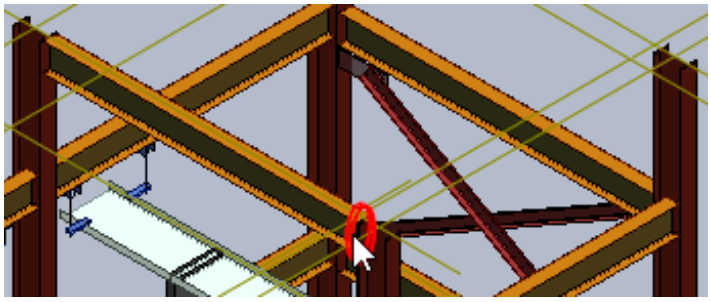
Move a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to move.
4. Click **Move** on the main ribbon.
5. Define the first point of a vector used to move the member.
6. Define the second point of the vector.

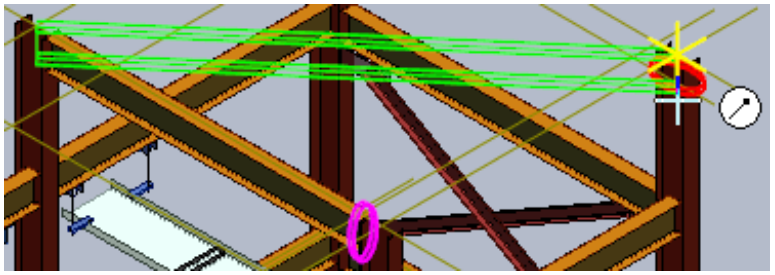
NOTE The frame connections might prevent you from moving the member as you would like. For example, a supported member has a seated frame connection to a supporting member. The seated frame connection prevents you from moving the supported member off of the supporting member. You can, however, slide the supported member along the supporting member as long as the seated frame connection is still valid. Consider copying and pasting a member that you want to move, and then deleting the original.

Move one end of a member

1. Click **Select** on the vertical toolbar.
2. Select **Frame Connections** in the **Locate Filter**.
3. Select the frame connection that is near the member end to move.



4. In the **Connection** option, select **By Rule**.
5. Define the new location for the member end.



Member System Prismatic Properties Dialog Box

Specifies the properties for the standard member or designed member that you are editing. For an explanation of the difference between a standard member and a designed member, see *Members* (on page 22).

Member System Tab (Member System Prismatic Properties Dialog Box) (on page 61)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

Member System Tab (Member System Prismatic Properties Dialog Box)

Specifies the properties for the member system.

Category

Select the properties to view for the member system.

Standard

Name

Displays the name of the member system. The member system name is based on the **Name Rule** selection. If you want to type a new name for the member system, in the **Name Rule** box, select **User Defined**, and then type a name for the member system in the **Name** box. If you change the **Type Category** of the member system after defining a custom name, the name resets to the default name for the new member type.

Name Rule

Specify the naming rule to use to name this member system.

MemberSystemTypeNameRule - Names the member system using this method: <member type>-<location>-<index number> where <member type> is the selected **Type** property, <location> is the global workshare location ID, and <index number> is a unique index number that starts at 0001. For example: Purlin-1-0003.

Default Name Rule - Names the member system using this method: MemberSystem-<location>-<index number> where <location> is the global workshare location ID and <index number> is a unique index number that starts at 0001. For example: MemberSystem-1-

0045.

Unique Name Rule - Names the member system using this method: <parent system>-MemberSystem-<location>-<index number> where <parent system> is the name of the parent system that the Member System belongs to, <location> is the global workshare location ID, and <index number> is a unique index number that starts at 0001. For example: Structure System-MemberSystem-1-0001.

User Defined - Select this name rule to name the member system yourself using the **Name** box.

Parent System

Select the system to which the member system that you are placing belongs.

Type category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Priority

Specifies the priority assigned to the object system. Smart 3D uses the priority to group objects. To change the options on the list, edit the **Structural Member Priority** select list in Catalog.

Continuity Type

Indicates the way in which the object system reacts when the system intersects another object system (your automatic splitting preference). The list is defined by the StructContinuity codelist.

Continuous indicates that the object system should split the other object system.

Intercostal indicates that the object system should be split by the other object system.

You cannot split objects that have a **Continuity Type** setting of **Continuous**.

Continuity Priority Number

Specifies the priority when two profile systems that have the same **Continuity** setting intersect. The profile system with the lowest continuity priority number penetrates the other system. For example, two profile systems intersect and the **Continuity** setting of each set to **Continuous**. The profile system with the lowest **Continuity Priority** number splits the other profile system. You can only modify this property at the root system.

Align

Indicates whether the software copies offsets from the frame connection at the member system end to the unsupported frame connection at the other member system end.

True indicates that Smart 3D copies the offsets.

False indicates the Smart 3D does not copy the offsets.

Start East / **Start X**

Displays the X-coordinates of the start of the member relative to the active coordinate system.

Start North / **Start Y**

Displays the Y-coordinates of the start of the member relative to the active coordinate system.

Start Elevation / **Start Z**

Displays the Z-coordinates of the start of the member relative to the active coordinate system.

End East / **End X**

Displays the X-coordinates of the end of the member relative to the active coordinate system.

End North / **End Y**

Displays the Y-coordinates of the end of the member relative to the active coordinate system.

End Elevation / **End Z**

Displays the Z-coordinates of the end of the member relative to the active coordinate system.

Relationship Tab

Displays all objects related to the selected object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Additional examples for marine relationships are as follows:

- For plate and profile system properties, the related bounded objects, bounding objects, and connections are shown.
- For plate and profile system part properties, parent systems are shown.
- For assembly connection properties, all connected objects are shown.
- For the properties of a frame connection on a member, supported, supporting, and auxiliary supporting parts are shown.
- For split connection properties, the parent and auxiliary supporting parts are shown.

Name

Specifies the name of the object.

Type

Specifies the type of object. To change the options on the list, edit the **Weld Type** select list in Catalog.

Go To

Displays the properties of the selected object.

Configuration Tab

Displays the creation, modification, and status information about an object.

NOTE You cannot define the filters using the **Configuration** tab.

Plant

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

NOTE The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

NOTE You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified

Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

Transfer Ownership Dialog Box

Allows you to specify a new location and permission group for the selected model objects.

Current location

Displays the name of the location with which the current permission group is associated. All of the objects in the select set must belong to the same location.

Current permission group

Displays the name of the permission group with which the selected objects are currently associated. If all of the objects in the select set do not belong to the same permission group, this box appears blank.

New location

Specifies the name of the location to which you want to assign the objects. In a global workshare configuration, this box lists all the locations in which you have write access to one or more permission groups. The selection in this box filters the entries in the **New permission group** box.

New permission group

Specifies the new permission group to which to assign the selected objects. If you specify a value in the **New location** box, this list displays all permission groups to which you have write access in the selected location. If you do not specify a value in the **New location** box, this list includes all permission groups to which you have write access in all locations except the current location. This box is blank if you do not have write access to any permission groups at any locations other than the current one.

NOTE We strongly recommend that administrators follow naming convention rules that include the location as a prefix in the permission group name.

Notes Tab

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

NOTE Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, then only one of the notes shows on the drawing. It is important to know about and to consider this situation when defining notes on an object in the modeling phase. For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

Key point

Specifies the key point on the object to which you want to add a note.

Notes at this location, listed by name

Lists all notes for the selected key point on the object.

Date

Displays the date that the note was created. The system automatically supplies the date.

Time

Displays the time that the note was created. The system automatically supplies the time.

Purpose of note

Specifies the purpose of the note.

Author

Displays the login name of the person who created the note. The system automatically supplies this information. You cannot change this information.

Note text

Defines the note text. The software does not limit the length of the note text.

Show dimension

Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, then a dimension can be included for the component in the Support drawings, if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a requirement. The note **Name** and **Note text** are not used when you select this option.

New Note

Creates a new note on the object.

Standard Note

Displays a list of standard notes from which you can select. This feature is not available in this version.

Highlight Note

Highlights the note in the graphic view so that you can easily find the note and the object to which it is related. This feature is not available in this version.

Delete Note

Deletes the currently displayed note.

Member Part Prismatic Properties Dialog Box

Specifies the properties for the member part of the standard member that you are editing. For an explanation of the difference between a member system and a member part, see *Members* (on page 22).

Member Part Tab (Member Part Prismatic Properties Dialog Box) (on page 67)
Cross Section Tab (Member Part Prismatic Properties Dialog Box) (on page 73)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

Member Part Tab (Member Part Prismatic Properties Dialog Box)

Specifies the properties for the member.

Category

Select the properties to view for the member. Member part properties are divided into several different categories: **Standard**, **Weight and CG**, **Fabrication and Construction**, **Surface Treatment and Coating**, **Responsibility**, and **End Releases**. You select which category that you want to define values for by using the **Category** option.

Standard

Name

Displays the name of the member part. The member part name is based on the **Name Rule** selection. If you want to type a new name for the member part, in the **Name Rule** box, select **User Defined**, and then type a name for the member part in the **Name** box.

Name Rule

Specify the naming rule to use to name this member part.

- **Member Part Type Name Rule** - Names the member part using this method: <member type>-<location>-<index number> where <member type> is the selected **Type** property, <location> is the global workshare location ID, and <index number> is a unique index number that starts at 0001. For example: Beam-1-0003.
- **Default Name Rule** - Names the member part using this method: MemberPartPrismatic-<location>-<index number> where <location> is the global workshare location ID and <index number> is a unique index number that starts at 0001. For example: MemberPartPrismatic-1-0045.
- **Unique Name Rule** - Names the member part using this method: MemberPartPrismatic-<location>-<index number> where <location> is the global workshare location ID and <index number> is a unique index number that starts at 0001. For example: MemberPartPrismatic-1-0045.
- **User Defined** - Select this name rule to name the member system yourself using the **Name** box.

NOTE If you plan to export this model to Tekla Structures using CIMsteel, you must know that Tekla limits part names to 21 characters. If you use a Smart 3D naming rule that creates a member part name that is longer than 21 characters, Tekla truncates the name

during the CIMSteel import into Tekla. This truncation can cause all your member parts to have the same name in Tekla because Smart 3D naming rules have the unique index at the end of the name.

Parent System

Specifies the name of the parent system. You can define new systems in the Systems and Specifications task.

Type category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Priority

Specifies the priority assigned to the object system. Smart 3D uses the priority to group objects. To change the options on the list, edit the **Structural Member Priority** select list in Catalog.

Length

Displays the length of the member without cutbacks applied. You cannot change this value.

Cut Length

Specifies the length of the member with cutbacks applied. You cannot change this value. A cutback is that part of a member removed by an assembly connection or by a manually placed trim (a cope, for example).

Reporting Requirements

Specify whether this object is reported.

Reporting Type

Select the reporting requirements code for the object. Valid codes are defined in Catalog in the **Reporting Type** select list.

Piece Mark

Specifies the piece mark of the member. Piece marks are typically used when importing and exporting member through CIMsteel. If a piece mark displays, the mark is usually the identity of the manufactured part from the other software package.

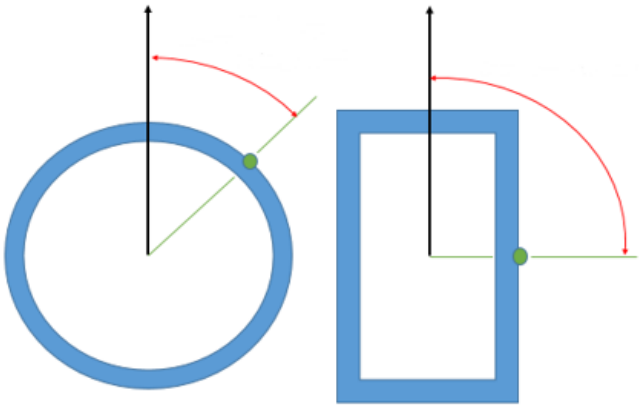
Assembly Mark

Specifies the assembly mark of the member. If the member was imported through CIMsteel, the assembly mark is usually the identity of the manufactured assembly to which this member belongs.

Seam Angle

Defines the location of the seam. This angle is measured clockwise from orientation vector

of the member looking down the member from the start end.



Seam Angle is only available on cross-sections that use the **Has Seam** definition. Refer to the *Smart 3D Symbol 2D User's Guide* and *Symbol 2D Reference Data Guide* for more information about cross-sections.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material density multiplied by the object's solid volume. Therefore, the material of the object affects the weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

NOTE For equipment, the **Weight and CG** property **Wet Weight** is the sum of **Dry Weight** and **Water Weight**. The dry weight and water weight values are catalog properties entered on the part sheet for the equipment.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Wet CG X

Specifies the X-axis location of the wet center-of-gravity.

Wet CG Y

Specifies the Y-axis location of the wet center-of-gravity.

Wet CG Z

Specifies the Z-axis location of the wet center-of-gravity.

Dry WCG Origin

Specifies the way in which the dry weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the dry weight center-of-gravity location relative to the active coordinate system.

Wet WCG Origin

Specifies the way in which the wet weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the wet weight center-of-gravity location relative to the active coordinate system.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Surface Treatment and Coating

Exterior Coating Requirement

Specifies the coating requirement for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Exterior Coating Area

Specifies the area of the coating for the object.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

End Releases

Start Member Release

Select the directions to release at the start of the member part. Directions are defined in the local coordinate system of the member system. If the combination of directions is not available, select the **User Defined** option and define the releases yourself.

Start X Displacement

Defines if the X direction at the start of the member part is fixed or free.

Start Y Displacement

Defines if the Y direction at the start of the member part is fixed or free.

Start Z Displacement

Defines if the Z direction at the start of the member part is fixed or free.

Start X Rotation

Defines if the X moment direction at the start of the member part is fixed or free.

Start Y Rotation

Defines if the Y moment direction at the start of the member part is fixed or free.

Start Z Rotation

Defines if the Z moment direction at the start of the member part is fixed or free.

End Member Release

Select the directions to release at the end of the member part. Directions are defined in the local coordinate system of the member system. If the combination of directions is not available, select the **User Defined** option and define the releases yourself.

End X Displacement

Defines if the X direction at the end of the member part is fixed or free.

End Y Displacement

Defines if the Y direction at the end of the member part is fixed or free.

End Z Displacement

Defines if the Z direction at the end of the member part is fixed or free.

End X Rotation

Defines if the X moment direction at the end of the member part is fixed or free.

End Y Rotation

Defines if the Y moment direction at the end of the member part is fixed or free.

End Z Rotation

Defines if the Z moment direction at the end of the member part is fixed or free.

See Also

Member Part Prismatic Properties Dialog Box (on page 67)

Cross Section Tab (Member Part Prismatic Properties Dialog Box)

Specifies the properties for the cross-section of a standard member part.

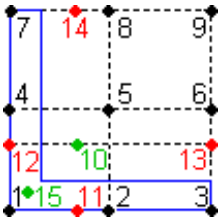
Section Standard

Specifies the section library from which you selected the section name. Sections are defined in the reference data.

Member Section Name

Defines the cross-section for the member. If you know the section name, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See Structure Reference Data Guide for more information about reference data.

Cardinal Point

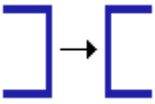


Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

Defines the angle by which the section is rotated about the member axis.

Reflect



Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.

Material

Select a material for the member. Materials are defined in the **AllCommon.xls** workbook or in the Catalog task.

Grade

Select a material grade for the member. Material grades are defined in the **AllCommon.xls** workbook or in the Catalog task.

Centroid X

Displays the location of the centroid along the local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Centroid Y

Displays the location of the centroid along the local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding

workbook.

Moment of Inertia about X (Ixx)

Displays the moment of inertia for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about Y (Iyy)

Displays the moment of inertia for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Statical Moment (Sw)

Displays the warping statical moment. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about X (Sxx)

Displays the section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about Y (Syy)

Displays the section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Torsional Moment of Inertia (J)

Displays the torsional moment of inertia for the section. This property is read- only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Constant (Cw)

Displays the warping constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Flexural Constant (H)

Displays the flexural constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Polar Radius of Gyration about Shear Center (ro)

Displays the polar radius of gyration about the shear center. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about X axis (Rxx)

Displays the radius of gyration for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Y axis (Ryy)

Displays the radius of gyration for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the

corresponding workbook.

Radius of Gyration about Principle XY (Rxy)

Displays the radius of gyration about the principle xy-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about X (Zxx)

Displays the plastic section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about Y (Zyy)

Displays the plastic section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Is Hollow

Displays **True** if the cross-section is hollow, such as a tube or can. Displays **False** if the cross-section is solid, such as a bar, or open, such as an I-beam.

Is Symmetric About X

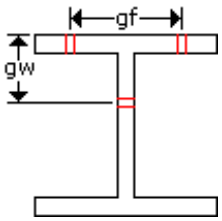
Displays **True** if the cross-section is symmetric about the local x-axis of the cross-section.

Is Symmetric About Y

Displays **True** if the cross-section is symmetric about the local y-axis of the cross-section.

Flange Gage (gf)

Displays the bolt gage for the flange. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.



Web Gage (gw)

Displays the bolt gage for the web. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Section Name

Specifies the section name. This name displays when you label members. This property is read-only. To edit this value, edit the cross-section's properties in Catalog or in the corresponding workbook.

Short Name

Displays the short name for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Alternate EDI Name

Displays the Electronic Data Interchange name for the section. This name is used when translating sections through CIMsteel. This property is currently not used.

Description

Specifies the description. This property is read-only. To edit this value, you need to edit the

cross- section's properties in Catalog or in the corresponding workbook.

Area

Specifies the cross-section area for the object. This property is read-only. To change this value, edit the object properties in Catalog or in the corresponding workbook.

Depth

Displays the depth for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Width

Displays the flange width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Perimeter

Displays the outside perimeter distance for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Unit Weight

Displays the weight of the section. The unit weight is defined in mass per length pound per foot (lbm/ft). This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Theoretical Maximum Yield Stress (Fy''')

Displays the maximum yield stress for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Statical Moment at Point in Flange (Qf)

Displays the first moment of area for the flange. This property is read- only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Statical Moment at Mid Depth of Section (Qw)

Displays the first moment of area for the web. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Normalized Warping Function (Wno)

Displays the normalized warping function. The function is defined in square inches or square millimeters. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Beam Buckling Factor (X1)

Displays the beam buckling factor. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Beam Buckling Factor (X2)

Displays the beam buckling factor. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Web Thickness (tw)

Displays the web thickness for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Web Depth (d)

Displays the web depth. This property is read-only. To edit this value, you need to edit the cross- section's properties in the Catalog task or in the corresponding workbook.

Flange Thickness (tf)

Displays the flange thickness for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Flange Width (bf)

Displays the width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Distance to Web Toe Fillet (kdetail)

Displays the distance from the outer face of the flange to the web toe of the fillet of the rolled shape or the equivalent distance on welded section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Distance to Web Toe Fillet (kdesign)

Displays the distance from the outer face of the flange to the web toe of the fillet of the rolled shape or the equivalent distance on the welded section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Group Id

Displays the material group identification for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Designed Member Properties Dialog Box

Specifies the properties for the designed member that you are editing. For an explanation of the difference between a standard member and a designed member, see *Members* (on page 22).

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

Occurrence Tab (Designed Member Properties Dialog Box) (on page 78)
Cross Section Tab (Designed Member Properties Dialog Box) (on page 82)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

Occurrence Tab (Designed Member Properties Dialog Box)

Specifies the properties for the select designed member.

Category

Select the properties that you want to view for the member. Only **Standard** properties are available for designed members.

Show Dimensional Legend

Displays a graphic indicating the different dimensions that can be specified for the selected designed member.

Standard

Name

Displays the name of the designed member. The designed member name is based on the **Name Rule** selection. To type a new name for the designed member, select **User Defined** in the **Name Rule** box, and then type a name for the designed member in the **Name** box.

Name Rule

Specify the naming rule to use to name this designed member. You can select one of the listed rules or select **User Defined** and type the designed member name in the **Name** box.

Parent System

Specifies the name of the parent system. You can define new systems in the Systems and Specifications task.

Section Standard

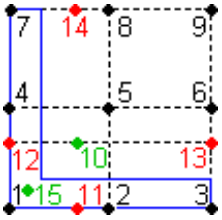
Specifies the section library from which you selected the section name. Sections are defined in the reference data.

Member Section Name

Defines the cross-section for the member. If you know the section name that you want, you can type it. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type BUIH* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. For more information about reference data, see *Structure Reference Data Guide*.

Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

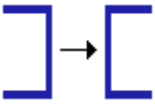


Angle

Defines the angle by which the section is rotated about the member axis.

Reflect

Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option is when you want the flanges of a channel section to point in the opposite direction.



Type category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Priority

Specifies the priority assigned to the object system. Smart 3D uses the priority to group objects. To change the options on the list, edit the **Structural Member Priority** select list in Catalog.

Centroid X

Displays the location of the centroid along the local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Centroid Y

Displays the location of the centroid along the local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about X (Ixx)

Displays the moment of inertia for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about Y (Iyy)

Displays the moment of inertia for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Statical Moment (Sw)

Displays the warping statical moment. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about X (Sxx)

Displays the section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about Y (Syy)

Displays the section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Torsional Moment of Inertia (J)

Displays the torsional moment of inertia for the section. This property is read- only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Constant (Cw)

Displays the warping constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Flexural Constant (H)

Displays the flexural constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Polar Radius of Gyration about Shear Center (ro)

Displays the polar radius of gyration about the shear center. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about X axis (Rxx)

Displays the radius of gyration for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Y axis (Ryy)

Displays the radius of gyration for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Principle XY (Rxy)

Displays the radius of gyration about the principle xy-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about X (Zxx)

Displays the plastic section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about Y (Zyy)

Displays the plastic section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Is Hollow

Displays **True** if the cross-section is hollow, such as a tube or can. Displays **False** if the cross-section is solid, such as a bar, or open, such as an I-beam.

Is Symmetric about X

Displays **True** if the cross-section is symmetric about the local x-axis of the cross-section.

Is Symmetric about Y

Displays **True** if the cross-section is symmetric about the local y-axis of the cross-section.

Area

Specifies the cross-section area for the object. This property is read-only. To change this value, edit the object properties in Catalog or in the corresponding workbook.

Depth

Displays the depth for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Width

Displays the flange width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Perimeter

Displays the outside perimeter distance for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Cross Section Tab (Designed Member Properties Dialog Box)

Specifies the properties for the cross-section of a designed member.

Name

Displays the cross-section name for the designed member. Sections are defined in the reference data. For more information about reference data, see *Structure Reference Data Guide*.

Centroid X

Displays the location of the centroid along the local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Centroid Y

Displays the location of the centroid along the local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about X (Ixx)

Displays the moment of inertia for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about Y (Iyy)

Displays the moment of inertia for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Statical Moment (Sw)

Displays the warping statical moment. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about X (Sxx)

Displays the section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about Y (Syy)

Displays the section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Torsional Moment of Inertia (J)

Displays the torsional moment of inertia for the section. This property is read- only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Constant (Cw)

Displays the warping constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Flexural Constant (H)

Displays the flexural constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Polar Radius of Gyration about Shear Center (ro)

Displays the polar radius of gyration about the shear center. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about X axis (Rxx)

Displays the radius of gyration for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Y axis (Ryy)

Displays the radius of gyration for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Principle XY (Rxy)

Displays the radius of gyration about the principle xy-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about X (Zxx)

Displays the plastic section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about Y (Zyy)

Displays the plastic section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Is Hollow

Displays **True** if the cross-section is hollow, such as a tube or can. Displays **False** if the cross-section is solid, such as a bar, or open, such as an I-beam.

Is Symmetric About X

Displays **True** if the cross-section is symmetric about the local x-axis of the cross-section.

Is Symmetric About Y

Displays **True** if the cross-section is symmetric about the local y-axis of the cross-section.

Flange Gage (gf)

Displays the bolt gage for the flange. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Web Gage (gw)

Displays the bolt gage for the web. This property is read-only. To edit this value, you need to

edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Plastic Neutral Axis (xp)

Displays the distance from the designated edge of the cross-section's plastic neutral axis (PNA) along the local x-axis.

Plastic Neutral Axis (yp)

Displays the distance from the designated edge of the cross-section's plastic neutral axis (PNA) along the local y-axis.

Section Name

Specifies the section name. This name displays when you label members. This property is read-only. To edit this value, edit the cross-section's properties in Catalog or in the corresponding workbook.

Area

Specifies the cross-section area for the object. This property is read-only. To change this value, edit the object properties in Catalog or in the corresponding workbook.

Depth

Displays the depth for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Width

Displays the flange width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Perimeter

Displays the outside perimeter distance for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Unit Weight

Displays the weight of the section. The unit weight is defined in mass per length pound per foot (lbm/ft). This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Nonlinear Plate System Properties Dialog Box

Specifies the properties for the root or leaf plate system of the designed member that you are editing. A designed member contains multiple plate systems that you can individually edit.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

NOTES

- A leaf system always shares the same property values as its root system upon creation.
- When a root and leaf system have properties with the same values, a change to a root system value changes the leaf system value.
- When you change the value of a property of a leaf system, later changes to that property on the root system do not change the leaf system value.
- When a root system is split, new leaf systems are created that inherit the property values of the root system, *except* for values on the original leaf system that are changed before the split.
- Some leaf system properties cannot be changed at the leaf system level. They must be changed at the root system level.

Main Tab (Nonlinear Plate System Properties Dialog Box) (on page 85)
Material Tab (Nonlinear Plate System Properties Dialog Box) (on page 88)
Molded Conventions Tab (Nonlinear Plate System Properties Dialog Box - Designed Member) (on page 89)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
General Tab (on page 94)

Main Tab (Nonlinear Plate System Properties Dialog Box)

Specifies the general properties of the nonlinear extruded plate system.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the plate system.

StdPlateSystemNamingRule - Uses syntax based on the reference plane.

- Plate system on a reference plane: *<Reference plane name>-<Index number><Plate type>-<Workshare location ID>*. For example, F383-1TBH-1, where F383 is the reference plane name, 1TBH is an index number appended by the Plate Type (Transversal Bulkhead in this example), and the final 1 is the Workshare Location ID. Child plates inherit the parent name and add a unique index number to the end, such as F383-1TBH-1-103.
- Plate system not created on a reference plane, or on a standalone plane (created from a copy or model data reuse operation): *<Global CS axis><Global CS position along the axis in mm>-<Index number><Plate type>-<Workshare location ID>*, as shown in the following examples.

  Global CS Location: 2 m
  XY Plane at Global Z Location: Z2000-1DCK-1
  ZX Plane at Global Y Location: Y2000-1LBH-1

YZ Plane at Global X Location: Y2000-1TBH-1

Global CS Location: 2.75 m
XY Plane at Global Z Location: Z2750-1DCK-1
ZX Plane at Global Y Location: Y2750-1LBH-1
YZ Plane at Global X Location: Y2750-1TBH-1

- Plate system not orthogonal to any major plane: *A-<Unique index number><Plate type>-<Workshare location ID>*, such as A-202DCK-1.
- Standalone planes used as boundaries, created from a copy or model data reuse operation: *Plane: <Global CS axis> = <Global CS position along the axis>*, such as Plane: Z = 2500mm. The name only displays in the **Boundary List** Dialog Box.

StdHierarchyChildNamingRule - Uses the following syntax: *<Parent system name>-<Object type>*. For example, Model_MDB-IJPlate1, where Model_MDB is the parent system, IJPlate is the object type, and 1 is an index number appended to the object type.

User Defined - Allows you to type any name.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Specifies a parent system for the plate system. You can define parent systems in the Systems and Specifications task. When you create a plate system, the software uses the property values of the parent system as the initial property values for the plate system. When a parent property value changes, the corresponding child property value also updates.

If this plate system is a bracket system, you cannot select another root plate system as the parent system because a bracket system is a root plate system.

Surface Geometry Type

Displays the Molded Forms command used to create the plate system.

Specification

Defines the structural specification for the plate system. This property can only be modified

at the root system.

Description

Defines an optional description for the plate system.

Continuity

Specifies the continuity type for the plate system. Continuity defines how the plate system reacts when it intersects another plate or profile system. Select **Continuous** to indicate that the plate system penetrates the other system. Select **Intercostal** to indicate that the plate system is penetrated by the other system. This property can only be modified at the root system.

Split Priority

Specifies the continuity priority. This priority is used to specify which plate system is continuous and which penetrated (split) when two plate systems intersect, but have the same value for **Continuity**. Plate systems with a lower continuity priority (such as 1, 2, or 3) penetrate plate systems with a higher continuity priority (such as 7, 8, or 9). This property can only be modified at the root system.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Tightness

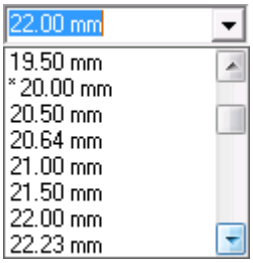
Specifies the level of tightness as it applies to the entire plate system.

Active

Determines whether a leaf system is active and applicable for modeling, drawing, and reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* in the *Molded Forms User's Guide* or in the *Common User's Guide*.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Material Tab (Nonlinear Plate System Properties Dialog Box)

Specifies the material properties for the nonlinear extruded plate system that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

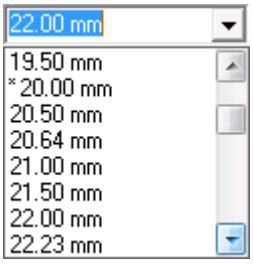
Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:



This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Molded Conventions Tab (Nonlinear Plate System Properties Dialog Box - Designed Member)

Specifies the plate orientation rules for the plate system of the designed member that you are editing.

Plate System Type

Displays the plate system type specified on the **Main Tab**. The delivered reference data provides different orientation settings for each type. A plate on a designed member can be **Flange Plate**, **Web Plate**, or **Tube Plate**. Plate system types are defined by the designed member cross-section. You cannot change the type.

Flange Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member. This is the default value for designed members.



Out - Positions thickness away from the center of the member.



Up - Positions thickness in the positive v-axis direction.



Down - Positions thickness in the negative v-axis direction.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the member. This is the default value for designed

members.



Out - Orients profiles away from the center of the member.



Down - Orients profiles in the negative v-axis direction.



Up - Orients profiles in the positive v-axis direction.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run parallel to the length of the designed member

In - Orients profile flanges towards the member plate web. This is the default value for designed members.



Out - Orients profile flanges away from the member plate web.



Left - Orients profile flanges in the negative u-axis direction.



Right - Orients profile flanges in the positive u-axis direction.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed

member.



Web Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member.



Out - Positions thickness away from the center of the member.



Left - Positions thickness in the negative u-axis direction.



Right - Positions thickness in the positive u-axis direction.



Centered - Centers thickness about the molded surface. This is the default value for designed members.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Out - Orients profiles away from the center of the member.



In - Orients profiles towards t center of the member.



Left - Orients profiles in the negative u-axis direction.



Right - Orients profiles in the positive u-axis direction. This is the default value for designed members.



Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the member plate web. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the member plate web.



Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed member.



Tube Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the tube member.



Out - Positions thickness away from the center of the tube member. This is the default value for designed tube members.



Centered - Centers thickness about the molded surface.



Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed tube members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the tube member.



Out - Orients profiles away from the center of the tube member. This is the default value for designed tube members.



Profiles Secondary Orientation

Ring Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed tube member. This is the default value for designed members.



Towards End - Orients profile flanges in the direction of the end point of the designed tube member.



Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the plate of a designed tube member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the axis of the designed tube member. This is the default value for designed members.



Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the axis

of the designed tube member.



General Tab

The **General** tab displays the properties that were selected by you or automatically determined by the software at creation. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

The properties displayed duplicate those available on other tabs, such as the **Main** tab. You can edit the properties on these tabs.

Plate Part Properties Dialog Box

Specifies the properties for the plate part that you are editing.

Main Tab (Plate Part Properties Dialog Box) (on page 94)
Material Tab (Plate Part Properties Dialog Box) (on page 95)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
General Tab (Plate Part Properties Dialog Box) (on page 96)
Weight & CG Tab (on page 98)
Extended User Attributes Tab (on page 98)
Routing Tab (on page 98)

Main Tab (Plate Part Properties Dialog Box)

Specifies the general properties of the plate part.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the object.

PlatePartRule - Creates a name based on the parent root plate system name, the location within the parent root plate system, and the parent assembly name.

User Defined - Name the plate part yourself by using the appropriate box.

Type

Displays the type of the plate part. It is inherited from the parent system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Displays the parent system for the object.

Surface Geometry Type

Displays the Molded Forms command used to create the root parent plate system. It is inherited from the parent system.

Specification

Specifies the specification for the object. This property is inherited from the parent system.

Description

Specifies a description for the object.

Tightness

Specifies the water tightness of the object. It is inherited from the parent system.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Material Tab (Plate Part Properties Dialog Box)

Displays the material properties for the plate part that you are editing. The material properties are inherited from the parent system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Actual Thickness

Displays the material thickness for the object.

General Tab (Plate Part Properties Dialog Box)

Specifies general properties of the plate part. Some properties displayed duplicate those available on other tabs, such as the **Main** tab. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

Category: Standard

NamingCategory

Displays the codelist value for **Naming Category**, defined on the **Main** tab. For more information, see *Main Tab (Plate Part Properties Dialog Box)* (on page 94).

PlateTightness

Specifies the water tightness of the plate part. The list is defined by the StructPlateTightness codelist.

PlateType

Specifies the type of the plate part. The list is defined by the StructPlateType codelist.

Thickening Technique Requested

Specifies the technique requested to thicken the plate part.

This property affects only the behavior of plate parts derived from systems. The property defaults to **Procedural** at creation. Changing the value of this property can affect the geometry of the part as described below.

Standalone plate parts initially display nothing for this property. You can change the value, but those changes do not change the part or its geometry. Smart 3D thickens all standalone plate parts, including collars, using procedural thickening.

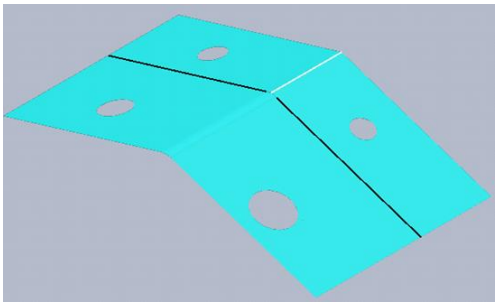
Undefined (blank) indicates that Smart 3D selects the appropriate technique. Smart 3D automatically changes this value to **Procedural**.

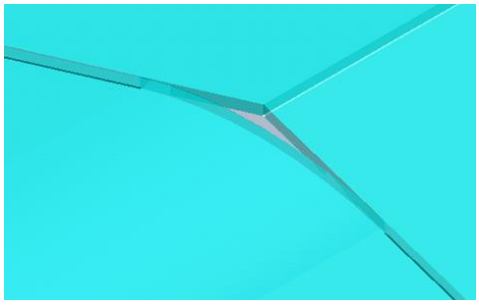
Procedural indicates the part is thickened using the standard thickening method provided by the ACIS modeler. This works for the vast majority of parts, and you should not change this value unless there is a specific reason to try a different technique.

NOTE If the thickening fails using ACIS, and the part is a hull part, the software automatically tries the **NonProcedural** thickening technique. This option only controls which method the software tries first. You can determine which method Smart 3D actually used from the **Thickening Technique Used** property.

Procedural with slit is used for parts that have internal slits or discontinuities in the surface of the system that form part of the boundary of parts on the system. The currently-supported example of this is a surface with a combination of split and bent knuckles joined end to end. Because Smart 3D does not support splitting knuckles and changing the properties of different segments, these surface must be constructed with the correct geometry to reflect the bent and split portions. Parts that are bounded by the slit, and potentially parts on the system that are within half a meter of the slit, must have this value set to assure a proper trim. Setting this attribute on parts that do not need it should not cause a failure but may cause the part to take more time to detail. The following pictures show an example of this type of surface, first the full surface and then a close-up in the region of the slit or discontinuity.

The plate system in these pictures is split into three parts by the knuckle (shown as white) and the two black seams. You must be sure that a single part does not touch both sides of the slit.





NonProcedural is an infrequently used option. Historically, a much higher percentage of parts failed to thicken with the ACIS modeler, and many designers used a specific hull form definition tool that produced a surface with very specific characteristics. **NonProcedural** thickening provided a tool that would thicken almost any part on this type of surface, even if the ACIS modeler failed. The ACIS modeler has improved greatly, and the **NonProcedural** thickening method has not been enhanced to handle the surfaces produced by many of the hull form tools currently in use. Setting this option may solve a failure-to-thicken problem on a hull part.

NOTE If **NonProcedural** thickening fails for a hull shell part, Smart 3D automatically tries to thicken the part using **Procedural** thickening. This option only controls which method the software tries first. You can determine which method Smart 3D actually used from the **Thickening Technique Used** property.

Thickening Technique Used

Displays the technique used to thicken the plate part. This is blank for a standalone plate part, although **Procedural** is always used for these parts. It displays one of the values defined for **Thickening Technique Requested** for a system-derived part. The meaning for these values is the same as defined for **Thickening Technique Requested**.

Category: Planning

Build Method

Specifies the method used to position child objects in the block. The list contains all available values as defined in the reference data for the assigned workcenter, typically **Vertical Drop**, **Drop at angle**, **Slide**, and **Default**. The list is defined by the BuildMethodData codelist.

Slot Connectivity

Specifies the minimum welding requirement at profile and slot intersections to meet the build method requirements. The list contains all available values as defined in the reference data, typically **None**, **Webleft**, **Webright**, **Double**, **N/A**, and **Default**. The list is defined by the SlotConnectivityData codelist.

Weight & CG Tab

Displays the center-of-gravity and the weight of the part.

Type

Displays the type of weight: **Dry Weight** or **Wet Weight**.

Weight

Displays the weight of the plate part pertaining to the weight type.

CoG X

Displays the position of center of gravity along the X-axis in the **CoG Coordinate System**.

CoG Y

Displays the position of center of gravity along the Y-axis in the **CoG Coordinate System**.

CoG Z

Displays the position of center of gravity along the Z-axis in the **CoG Coordinate System**.

CoG Coordinate System

Specifies the coordinate system to use to measure the center of gravity.

Extended User Attributes Tab

Displays the user attributes that have been bulkloaded on the report data object related to the part. Changes to these attributes do not cause the related manufacturing part to go out-of-date. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

Routing Tab

Specifies routing properties for the part. By default, no routing properties are set.

Create

Sets routing properties for the part.

Delete

Deletes the routing properties for the part.

Rule

Specifies a predefined rule set or a user-defined rule that is applicable to the part.

Workcenter

Select an assembly workcenter where the part is produced from a facility-defined rule set. The list contains all available workcenters regardless of their level in the workcenter hierarchy. Examples of workcenters include docks, shops, bays, and panel lines. Workcenters may also be areas and zones where a block is assembled from multiple assemblies.

Stage Code

Specifies the workcenter code.

No. Of Actions

Specifies the number of actions required to produce the part.

Action

Specifies the name of the action.

Machine

Specifies the type of machine used to perform the required operation.

Code

Specifies the machine code.

Frame Connection Properties Dialog Box

Specifies the properties for the frame connection that you are editing.

See Also

General Tab (Frame Connection Properties Dialog Box) (on page 99)
Frame Connections (on page 24)
Edit a Frame Connection (on page 57)
Relationship Tab (on page 63)
Notes Tab (on page 65)

General Tab (Frame Connection Properties Dialog Box)

Specifies the properties for the frame connection.

TIP The *supported* member is the member that you are placing. The *supporting* member is the existing member in the model to which you are connecting.

Category

Select the type of properties to view for the selected frame connection.

Type

Displays the type of frame connection. This box is read-only. See *Frame Connections* (on page 24) for more information about the different types that are available.

Property

Displays all properties associated with the selected frame connection. The list of available properties depends on what was defined in the reference data for the frame connection type.

Value

Specifies the values for the frame connection properties.

Name

Specifies the frame connection name.

Name rule

Specifies the naming rule used to name the frame connection.

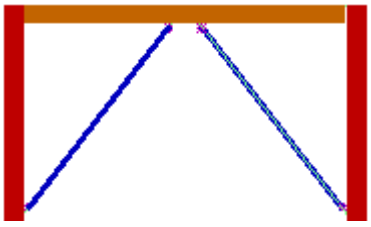
- **Default Name Rule** - Names the frame connection using the format "<frame connection type>-<location>-<index>" where <frame connection type> is what you select in the **Type** box on the ribbon, <location> is the global workshare location ID, and <index> is a unique index number that starts at 0001. For example, Axis-Along-1-0043.
- **User Defined** - Select this rule to name the frame connection yourself using the **Name** box.

Position rule

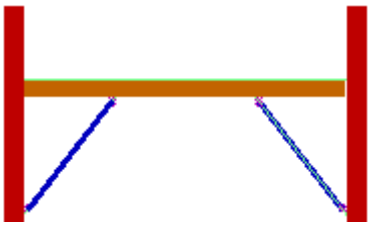
Defines how the frame connection is to behave when the supporting member is moved. You can select one of three options:

- **Intersection** - The member system lengthens or shortens to maintain the connections with the supporting member. The end of the supported member system slides to a new location on the supporting member.
- **Ratio** - The member system lengthens or shortens to maintain the connection with the supporting member. The end of the supported member system stays in the same relative position (that you can define) along the supporting member system. This option is similar to the **Distance** option except that you define a percentage ratio from the supporting member end.
- **Distance** - The member system lengthens or shortens to maintain the connection with the supporting member. The end of the supported member system stays in the same position (that you can define) along the supporting member system. This option is similar to the **Ratio** option except that you define an actual distance from the supporting member end.

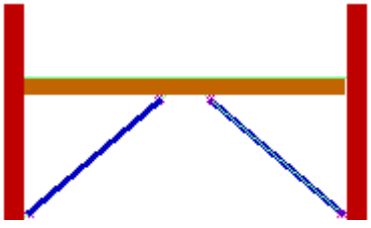
Original Position:



Intersection Position Rule:



Ratio and Distance Position Rule:



Distance along

Type the distance from the supporting member end that the supported member is positioned. You must include the units of measurement when defining this distance. The supporting member end that is measured from, is defined using the **End** option. This option is available when you set **Position Rule** to **Distance**.

Ratio

Type the ratio of the supporting member length that the supported member is positioned. For example, type **.25** if you want the supported member a fourth of the way along the supporting member. Type **.333** if you want the supported member a third of the way, and so forth. Which supporting member end that is measured from is defined using the **End** option. This option is available when you set **Position Rule** to **Ratio**.

End

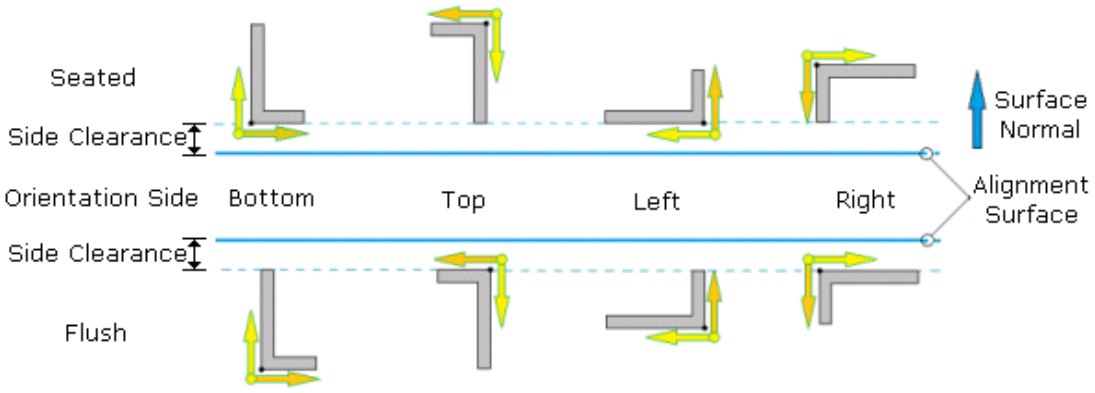
Specifies which end of the supporting member that the ratio or the distance-along distance is measured. You can select **Start**, **End**, or **Auto**. Start is the first member end that was placed. End is the second member end that was placed. If you select **Auto**, the software automatically selects the supporting member end that is closest to the frame connection. The **Auto** setting is recommended so that you do not have to worry whether the supporting member was modeled left-to-right, right-to-left, top-to-bottom, or bottom-to-top.

Align Frame Connection Properties

Side Justification

Select the side of the plane on which to place the member.

- **None** - Aligns the member axis so that it is placed on the surface.
- **Seated** - Sets the member on top of the surface. The selected **Orientation Side** is nearest the surface.
- **Center** - The center of the cross-section is placed on the surface. The center is calculated by the top-bottom, left-right bounds of the cross-section shape.
- **Flush** - Uses the alignment surface's top and bottom extent to position the supported member. The supported member typically lies within the body of the alignment plane with one edge of the member flush with the alignment surface but can be offset.



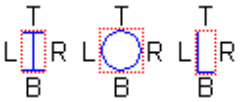
Orientation Control

If **On**, the member angle is controlled by the **Orientation Angle** value defined for the Align

frame connection. If **Off**, you can set the angle of the member independently of the align surface by using the **Rotation** property for the member part.

Orientation Side

Specifies the side of the supported member's cross-section that is mated to the alignment surface. You can specify **Top**, **Right**, **Bottom**, or **Left**. Edges of typical section shapes are shown in the figure.

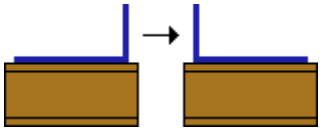


Orientation Angle

When **Orientation Control** is **On**, this angle is added to the selected **Orientation Side** to rotate the member. If the Align frame connection is used on both ends of the member and you define a different angle value for each end, the angle defined at the start end of the member is used.

Reflect

Reflects or mirrors the cross-section of the supported member about a plane perpendicular to the supporting member side. An example of when to use this option is when you place a supported member with an angle cross-section using the left edge option and you want the angle facing the other direction.



Extend Distance

Specifies the offset distance from the bounding surface along the member's axis.

Side Clearance

Specify the offset distance perpendicular to the align surface to place the member axis.

Lateral Distance

Specifies the offset distance in the plane of the align surface to place the member axis.

Axis Frame Connection Properties

X Offset

Specifies an offset to apply in the x-direction after the two cardinal points are aligned.

Y Offset

Specifies an offset to apply in the y-direction after the two cardinal points are aligned.

Z Offset

Specifies an offset to apply in the z-direction after the two cardinal points are aligned.

Coordinate System

Specifies the coordinate system to use for the offset values. Local is the local coordinate system of the supporting member.

Supporting CP

Specifies to which cardinal point on the supporting member system to align the supported member system's cardinal point. You can specify any cardinal point number, or select 0 to use the cardinal point with which the supporting member was placed.

Gap Frame Connection Properties

There are three members in a gap frame connection:

- The *target* member is the member always to move and is the owner of the frame connection. The target member is shown as blue in the figure below.
- The *primary* member is the member to which the other two members are attached. The primary member is shown as red in the figure below.
- The *secondary* member is the third member in the joint. The software does not require the secondary member to be in the same plane as the target member. The secondary member is shown as orange in the figure below.

Actual Gap Distance (G)

Displays the actual gap distance between the target and secondary member. This distance is shown as G in the figures below.

Actual Centerline Distance (X)

Displays the actual gap between the center line of the target member and the center line of the secondary member. This distance is shown as X in the figures below.

Actual Overlap Distance (L)

Displays the distance between the lowest point of the target and secondary member saddle on the primary member to the point where the target and secondary member intersect. This distance is shown as L in the figures below.

Actual Overlap Distance (A)

Displays the amount of overlap between the target and secondary member. This distance is shown as A in the figures below.

Allowable Centerline Distance

Displays the calculation of the primary member diameter times the **Centerline Design Factor**. The centerline distance should be less than or equal to this value.

Allowable Overlap Distance (Fmin × B)

Specify the minimum overlap distance (A) that is acceptable.

Allowable Overlap Distance (Fmax × B)

Specify the maximum overlap distance (A) that is acceptable.

Design Passed (Centerline)

Display **True** if the centerline design passed. Displays **False** if the centerline failed. The centerline design pass/fail is calculated by comparing actual centerline distance with the allowable centerline distance.

Design Passed (Overlap)

Display **True** if the overlap design passed. Displays **False** if the overlap failed. The overlap design pass/fail is calculated by (Min. Factor × B) < A < (Max. Factor × B) and then comparing that value to the **Allowable Overlap Distance (Fmin × B)** and the **Allowable Overlap Distance (Fmax × B)** values. You define the minimum and maximum design factors using the **Overlap Design Factor (Fmin)** and **Overlap Design Factor (Fmax)** properties. The B distance is shown in the figures below.

Direction

Select the direction to calculate the gap.

- **Axially Along Surface** - Select this mode when you want to define the offset as the distance between the center lines of the target and secondary members. This distance is measured parallel to the primary member centerline.
- **Axially and Flush Along Surface** - **This option is currently not available.** Select this mode when you want to define the offset as the distance between the center lines of the target and secondary members and you want the target member to be flush with the outside of the primary member. The axially distance is measured parallel to the primary member centerline. Use the **Flush Direction** and **Flush Offset** options to refine exact flush placement.
- **Radially Along Surface** - Select this mode when you want to define the offset as the distance between the closest points on the hull of the target and secondary members. This distance is measured around the hull of the primary member.

Offset Required

Type the gap distance that you want between the target and secondary member. Select how you want to define this distance using the **Offset Type** box.

Centerline Distance Roundoff

Type the distance to which the centerline gap should be rounded off.

Centerline Design Factor

Type the ratio of the primary member section size to use in calculating whether the centerline design passed or failed. The default value is **0.25**, which specifies that the primary member diameter be divided by 4 (D/4).

Flush Direction

This option is currently not available. Specify which direction you want the target member to move to become flush with the primary member.

Flush Offset

This option is currently not available. Specify the distance from the primary member hull edge to the target hull edge. An offset of zero indicates that the two member edges are perfectly flush.

Offset Along

Defines if the target member moves along the primary member or along the secondary member to create the gap. This property is only available for Gap Single Axial gap connections.

Offset Type

Select how you want to specify the **Offset Required** value.

- **Gap** - The **Offset Required** distance is between the two closest points of the target member hull and the secondary member hull. This is shown as G in the figures below.
- **Centerline** - The **Offset Required** distance is between the center line of the target member and the center line of the secondary member. This is shown as X in the figures below.

- **Overlap** - The **Offset Required** distance is between the lowest point of the target and secondary member saddle on the support member to the point where the target and secondary member intersect. This is shown as L in the figures below.

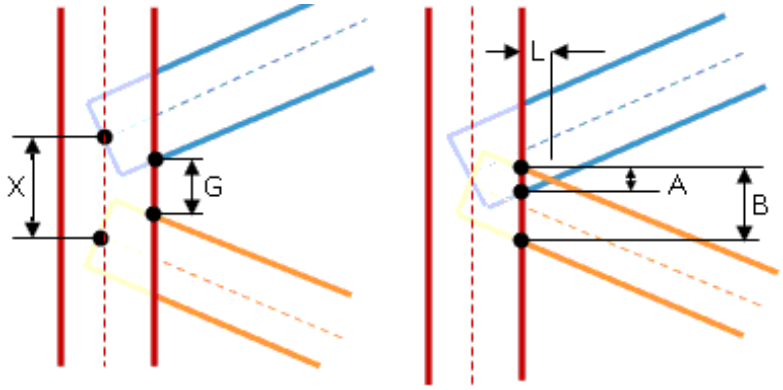
Overlap Design Factor (Fmin)

Defines the minimum design factor for the calculation: Min × B < A < Max × B. The default value is 0.3. This calculation is used to determine if **Design Passed (Overlap)** passes or fails. The A and B distances are shown in the figures below.

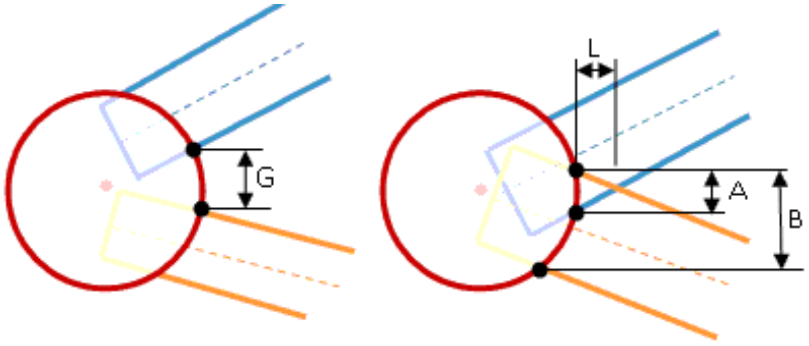
Overlap Design Factor (Fmax)

Defines the maximum design factor for the calculation: Min × B < A < Max × B. The default value is 0.5. This calculation is used to determine if **Design Passed (Overlap)** passes or fails. The A and B distances are shown in the figures below.

Axially Along Surface:



Radially Along Surface:



Seated, Flush, and Centerline Frame Connection Properties

Side

Select the side of the supporting member on which you want to place the supported member.

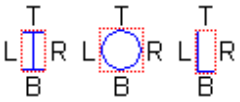
Offset

Specify the distance to place the supported member from the supporting member. For seated and flush frame connections, the offset is between the side of the supporting member that you specified with the **Side** option and the supported member's side that you specify with the **Edge** option. For centerline frame connections, the offset is between

centerline of the supporting member and the supported member's side that you specify with the **Edge** option.

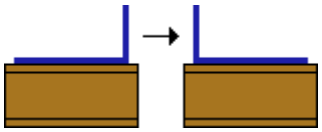
Edge

Specifies the side of the supported member's cross-section that is mated to the supporting member. You can specify **Top**, **Right**, **Bottom**, or **Left**. Edges of typical section shapes are shown in the figure.



Reflect

Reflects or mirrors the cross-section of the supported member about a plane perpendicular to the supporting member side. For example, when you place a supported member with an angle cross-section using the left edge option and you want the angle facing the other direction.



Surface Frame Connection Properties

X Offset

Specifies an offset to apply in the x-direction.

Y Offset

Specifies an offset to apply in the y-direction.

Z Offset

Specifies an offset to apply in the z-direction.

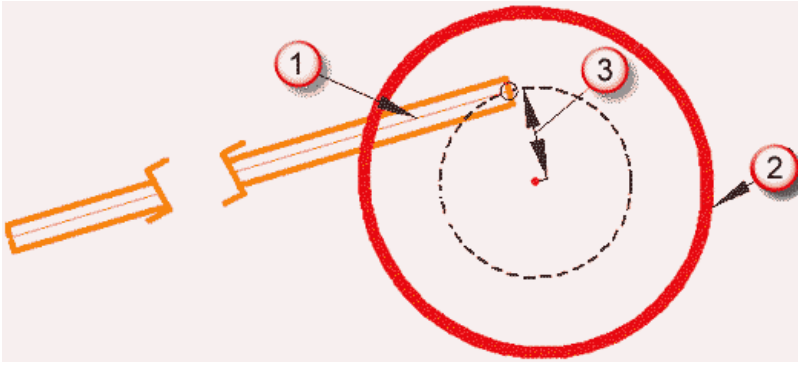
Coordinate System

Specifies the coordinate system to use for the offset values.

Tangent Frame Connection Properties

Offset Distance

Type the distance (**3**) from the centerline of the supporting member (**2**) to the tangent connection point. This distance is the radius of the circle to which the supported member (**1**) is made tangent.



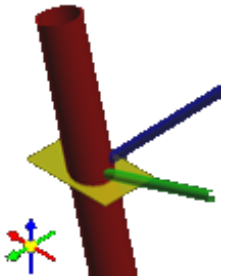
Side

Select **Left** or **Right** to define on which side of the supporting member the supported member is placed. To determine left and right, image you are standing on the start end of the member with your head towards positive global Z looking down the length of the member. Your left and right determine the direction of **Left** and **Right**.

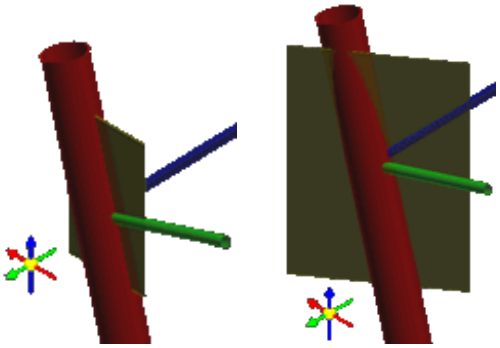
Tangent Plane

Controls the plane on which the tangent circle is drawn. You can select:

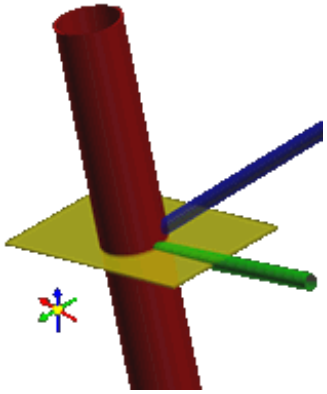
- **Horizontal** - The tangent circle is parallel to the X-Y plane (normal to global Z) regardless of the slope of any of the members. For example, you are creating a tangent connection between the sloping green beam and the sloping red column, or between the sloping blue beam and the sloping red column.



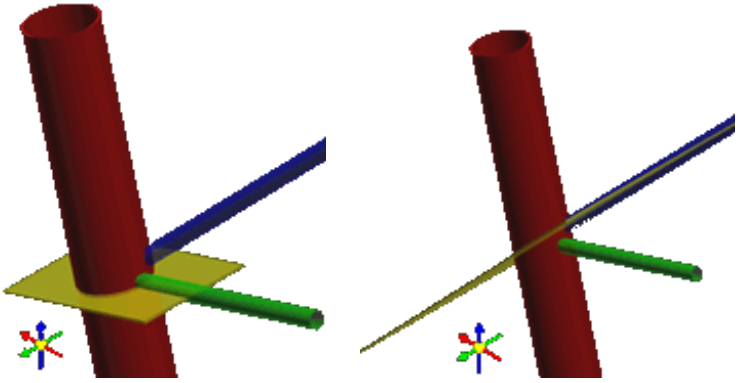
- **Vertical** - The tangent circle is parallel to global Z and is aligned with the centerline of the supported member (green member in the left figure, blue member in the right figure).



- **Normal to Supporting** - The tangent circle is normal to the supporting member (the sloping red column).



- **In Plane of Supported** - The tangent circle is parallel to the centerline of the supported member and normal to the supporting member centerline (green member in the left figure, blue member in the right figure).



Vertical Corner Brace Frame Connection Properties

X Offset

Specifies the offset to apply in the x-direction.

Y Offset

Specifies the offset to apply in the y-direction.

Z Offset

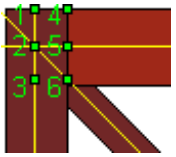
Specifies the offset to apply in the z-direction.

Coordinate System

Select whether the offset values are defined relative to the global coordinate system or the member's local coordinate system.

Work Point

Specifies the work point location. There are six work point locations that you can select.



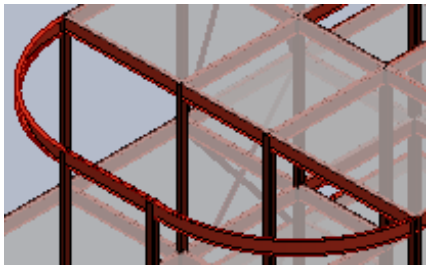
1 - Primary Center - Secondary Far Side
2 - Primary Center - Secondary Center
3 - Primary Center - Secondary Near Side
4 - Primary Near Side - Secondary Far Side
5 - Primary Near Side - Secondary Center
6 - Primary Near Side - Secondary Near Side

See Also

Frame Connection Properties Dialog Box (on page 99)

SECTION 4

Place Curve Members



Places a curved member in the model. You can place beams, columns, or braces member types using this command. For additional information, see *Members* (on page 22). Use this command when you want to place curved members by specifying the exact start and end points.

You can define custom member types by editing the **Structural Member Type** select list in the **Catalog** task.

NOTE You cannot place loads on curved members or define frame connection offsets for curved members.

Place Curve Member Ribbon

Specifies the properties for the curved member that you are placing.

Member Properties

Activates the **Member Properties** dialog box. You can use this dialog box to specify additional member properties, such as material and material grade, which you cannot set on the ribbon.

Path

Activates the Create Path Ribbon with which you define the curved member path.

Finish

Click to place the member in the model.

System

Select the system to which the member belongs. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see Select System Dialog Box.

Type Category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

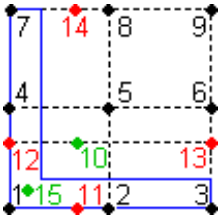
Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Section Name

Defines the cross-section for the member. If you know the section name, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See Structure Reference Data Guide for more information about reference data.

Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

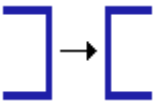


Angle

Defines the angle, in degrees or radians, by which the cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis depending on the member orientation.

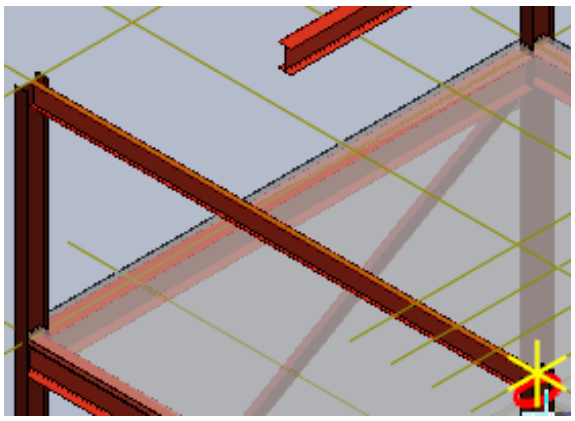
Reflect

Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.



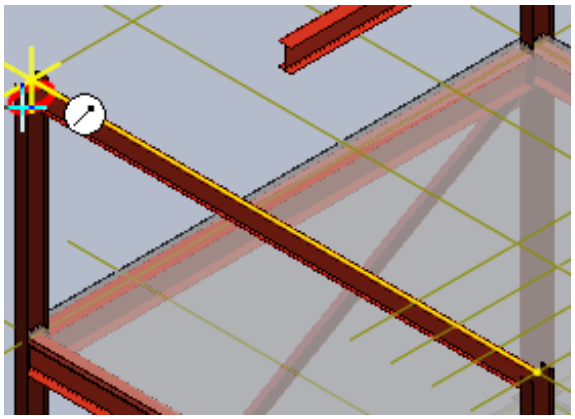
Place a curved member

1. Click **Place Curve Member** on the vertical toolbar.
2. Select an end location of the curved member.

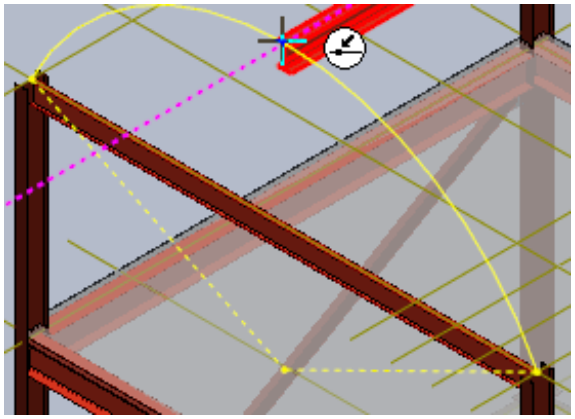


TIP You can select an existing member's frame connections to define the location, or define the location using a SmartSketch relationship such a Point Along.

3. Set **Path Type** to **Arc by End Points** .
4. Optionally, set the **Plane** option to match the plane of your curved member.
5. Select the opposite end location of the curved member.

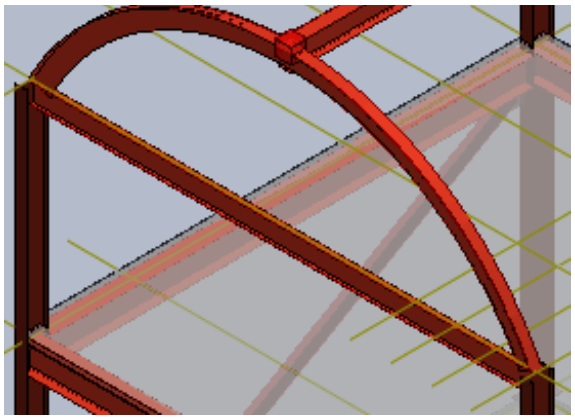


6. Select a point along the curved member or define the radius or sweep of the curved member.



7. Click **Finish**.
8. Select a system for the curved member.
9. Select a member type and type category for the member.
10. Select a section size and cardinal point.

11. Click **Finish**.



See Also

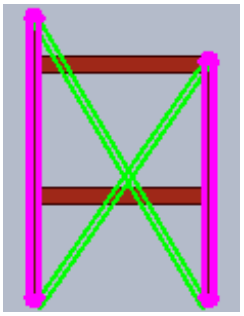
Define a Path (on page 501)
Sketching in 3-D (on page 493)

SECTION 5

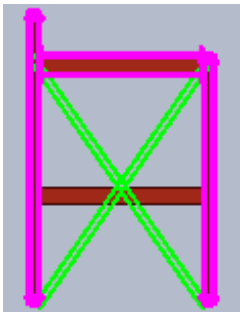
Place Bracing

Places cross bracing between members that you select in the model. You can place cross braces or chevron braces in a vertical plane, a horizontal plane, or any other non-orthographic plane. Although the two cross bracing members are placed at the same time, after placement the two members can be edited or deleted individually if needed. For additional information about members, see *Members* (on page 22).

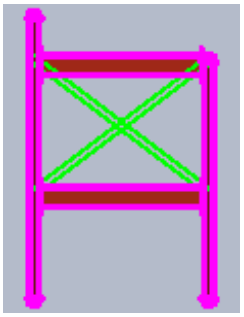
Vertical X Bracing



At a minimum, you must select two columns to define the cross bracing location. The two columns must be co-planar, but they do not need to be the same length or have the same starting or ending elevation. In this case, both ends of the braces are connected to the columns using the Axis-End frame connection. (For more information about frame connections, see *Frame Connections* (on page 24).) In addition, the cross brace members are always placed with the starting end of the brace member at the lower elevation.

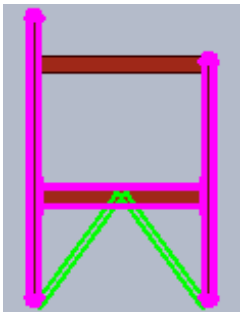


If you select two columns and a beam to define the cross bracing location, the bracing is placed at the intersection of the beam and the two columns to the two column bases. Again, the two columns must be co-planar, but they do not need to be the same length or have the same starting or ending elevation. In this case, the upper ends of the braces are connected to the columns and the beam using the Vertical Corner Brace frame connection. The bottom ends of the braces are connected to the column using the Axis-End frame connection. Again, the cross brace members are always placed with the starting end of the brace member at the lower elevation.

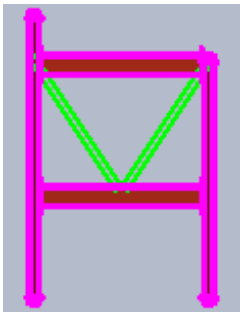


If you select two columns and two beams to define the cross bracing location, the bracing is placed in the intersection of the four members. Again, the two columns must be co-planar, but they do not need to be the same length or have the same starting or ending elevation. In this case, both ends of the braces are connected to the columns and the beams using the Vertical Corner Brace frame connection. Again, the cross brace members are always placed with the starting end of the brace member at the lower elevation.

Vertical Chevron Bracing

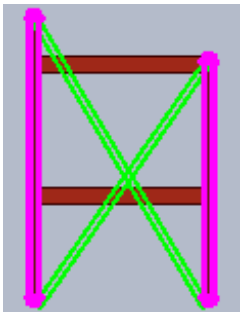


At a minimum, you must select two columns and one beam to define the vertical chevron bracing location. The two columns must be co-planar, but they do not need to be the same length or have the same starting or ending elevation. The chevron braces are connected to the columns using the Vertical Corner Brace frame connection. The beam mid-point is located to position the braces.

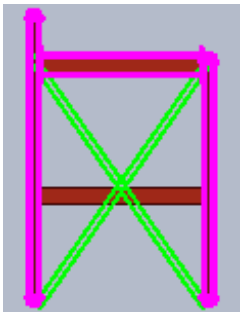


Optionally, you can select two columns and two beams to define the vertical chevron bracing location. The two columns must be co-planar, but they do not need to be the same length or have the same starting or ending elevation. The chevron braces are connected to the columns using the Vertical Corner Brace frame connection. The beam mid-point is located to position the braces.

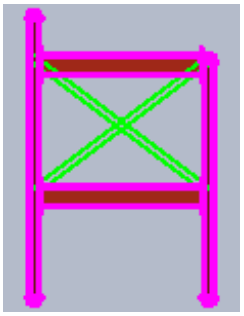
Horizontal X Bracing



At a minimum, you must select two members to define the cross bracing location. The two members must be co-planar, but they do not need to be the same length or have the same starting or ending location. In this case, both ends of the braces are connected to the select members using the Axis-End frame connection.

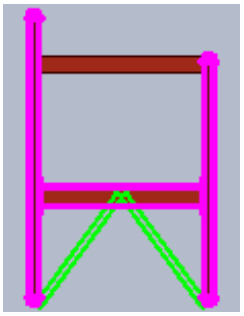


If you select two side members and a third member to define the cross bracing location, the bracing is placed at the intersection of the third member and the two side member to the two member ends. Again, the two side members must be co-planar, but they do not need to be the same length or have the same starting or ending locations. In this case, the upper ends of the braces are connected to the side members and the third member using the Vertical Corner Brace frame connection. The bottom ends of the braces are connected to the side members using the Axis-End frame connection.

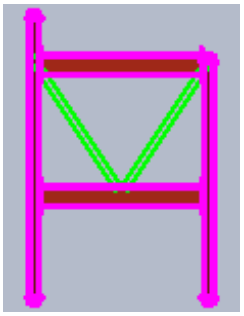


If you select two side members and a third and fourth member to define the cross bracing location, the bracing is placed in the intersection of the four members. Again, the two side members must be co-planar, but they do not need to be the same length or have the same starting or ending location. In this case, both ends of the braces are connected to the side members and the third and fourth members using the Vertical Corner Brace frame connection.

Horizontal Chevron Bracing



At a minimum, you must select two side members and one other member to define the point of the chevron bracing location. The two side members must be co-planar, but they do not need to be the same length or have the same starting or ending location. The chevron braces are connected to the side members using the Axis-End frame connection. The third member's mid-point is located to position the braces.



Optionally, you can select two side members and a third and fourth member to define the chevron bracing location. The two side members must be co-planar, but they do not need to be the same length or have the same starting or ending location. The chevron braces are connected to the side members using the Axis-End frame connection. The third member's mid-point is located to position the braces.

Place Bracing Ribbon

Specifies the properties for the cross braces members that you are placing or editing.

Member Properties

Activates the **Member Properties** dialog box. You can use this dialog box to specify additional member properties; such as material, material grade, and end releases; which you cannot set on the ribbon.

Select First Member

Select the first member that you want the cross bracing between. If you are placing vertical bracing, this member should be a column. If you are placing horizontal bracing, this member should be a beam.

Select Second Member

Select the second member that you want the cross bracing between. This member must be

coplanar with the first member that you selected.

Select First Beam

Select the first beam that you want the cross or chevron bracing between.

Select Second Beam

Select the second beam that you want the cross or chevron bracing between.

Finish

Click to place the cross or chevron bracing members in the model.

Bracing Type

Specifies the type of bracing to place. Select **Cross** to place two braces that cross. Select **Chevron** to place two braces as a chevron brace.

Connection

Select the frame connection type to use for the member that you are placing. If you select **By Rule**, the software determines the frame connection to use based on the geometry between the member that you are placing and existing members in the model. If you select **More**, all available frame connections display from which you can select the frame connection to use. For more information about frame connections, see *Frame Connections* (on page 24). This option is not available if you are editing an existing member. You can change the frame connection later by editing the frame connection directly.

System

Select the system to which the cross bracing members belong. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see "Select System Dialog Box" in *Place Linear Member Systems* (on page 35).

Type Category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

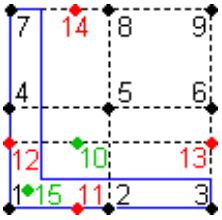
Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Section Name

Defines the cross-section for the member. If you know the section name, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See Structure Reference Data Guide for more information about reference data.

Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.



Angle

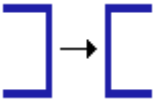
Defines the angle, in degrees or radians, by which the cross-section is rotated about the member axis. The zero-degree position is either the Z-axis or the X-axis of the active coordinate system depending on the member orientation. The active coordinate system is set using the **Coordinate System** control in **PinPoint** . The **Angle** property is not used when placing members using the **Seated**, **Flush**, or **Centerline** frame connections.

Offset

Defines the distance to place the supported members (the braces) from the supporting members.

Reflect

Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.



What do you want to do?

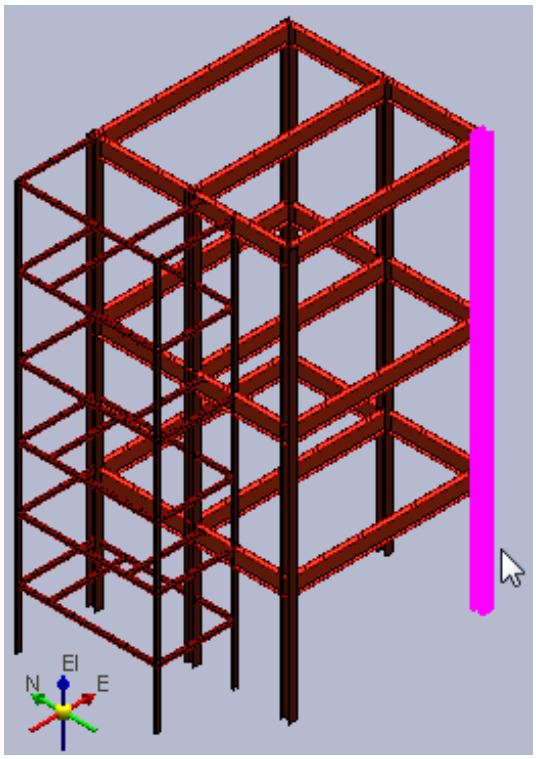
- *Place vertical cross bracing* (on page 117)
- *Place horizontal cross bracing* (on page 120)
- *Place vertical chevron bracing* (on page 121)
- *Place horizontal chevron bracing* (on page 124)

Place vertical cross bracing

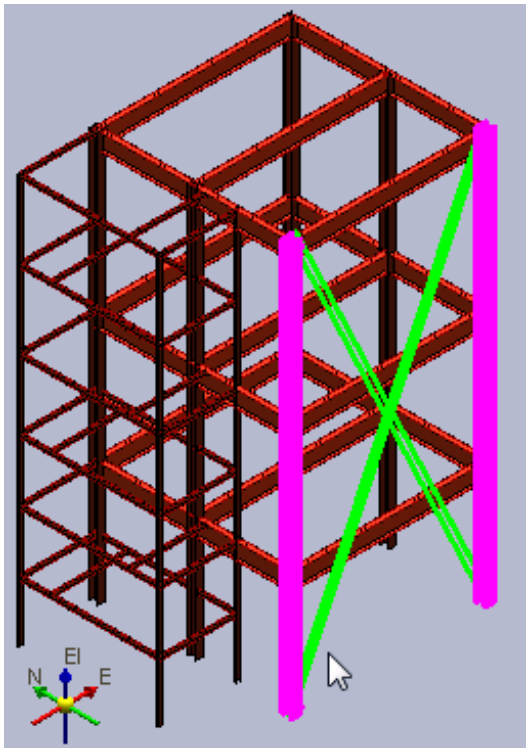
1. Click **Place X Bracing** on the vertical toolbar.
2. In the **Bracing Type** box, select **Cross**.
3. In the **Type category** box, select the cross bracing member type category to place.
4. In the **Type** box, select the cross bracing member type to place.
5. Set the **Section Name**, **Cardinal Point**, and **Angle** options as required.

   TIP You can set the beta angle relative to your rotated coordinate system by activating **PinPoint** and selecting the rotated coordinated system as the active coordinate system in the **Coordinate System** box.

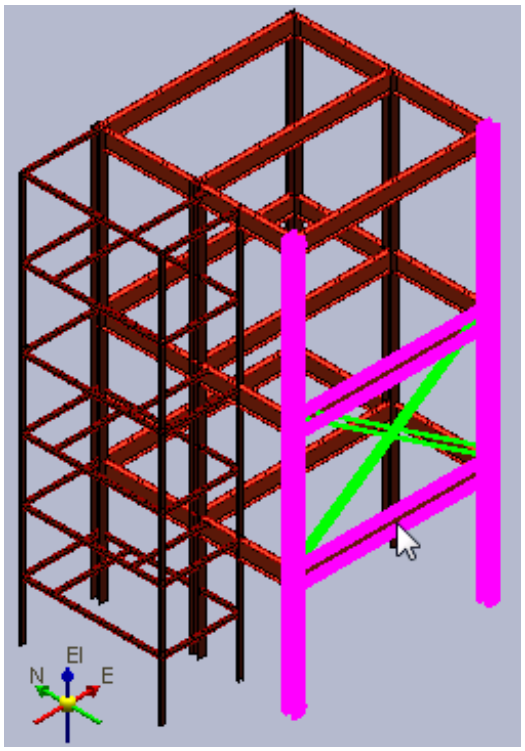
6. Select the first column to support the cross bracing.



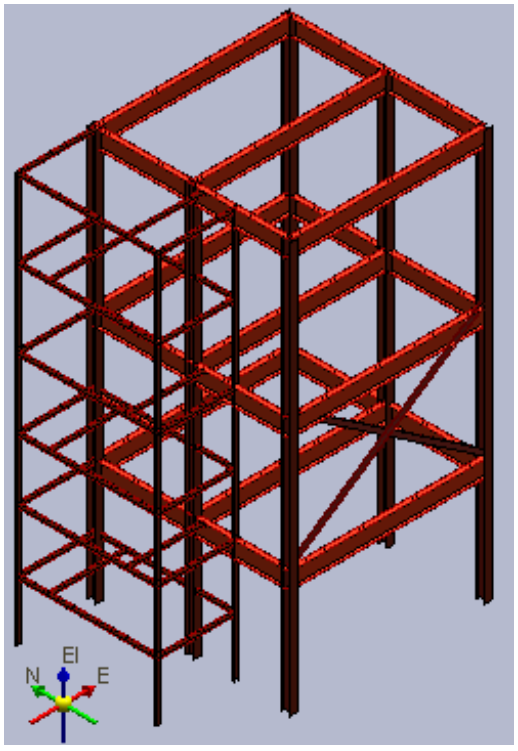
7. Select the second column to support the cross bracing.



8. Optionally, select one or two beams to support the cross bracing.



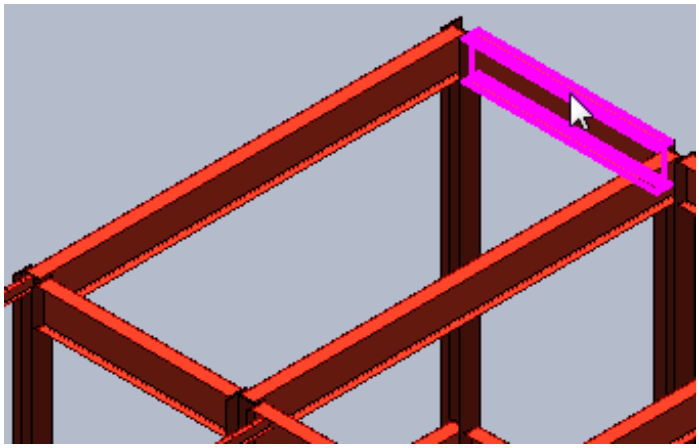
9. Click **Finish**.



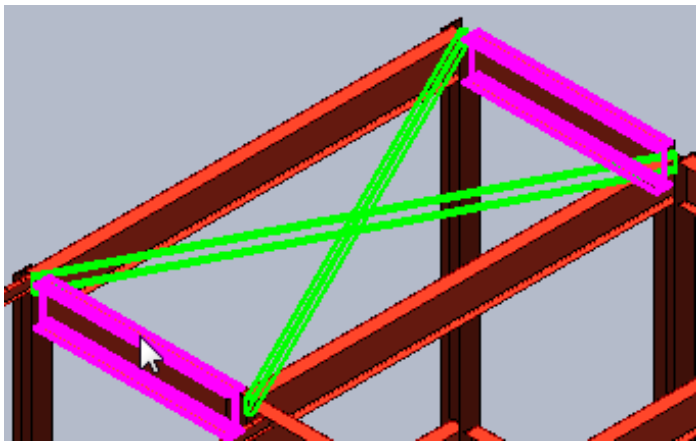
Place horizontal cross bracing

1. Click **Place X Bracing** on the vertical toolbar.
2. In the **Bracing Type** box, select **Cross**.
3. In the **Type category** box, select the cross bracing member type category to place.
4. In the **Type** box, select the cross bracing member type to place.
5. Set the **Section Name**, **Cardinal Point**, and **Angle** options as required.

   TIP You can set the beta angle relative to your rotated coordinate system by activating **PinPoint** and selecting the rotated coordinated system as the active coordinate system in the **Coordinate System** box.
6. Select the first member to support the cross bracing. This member is one of the "side" members to which the bracing connects.

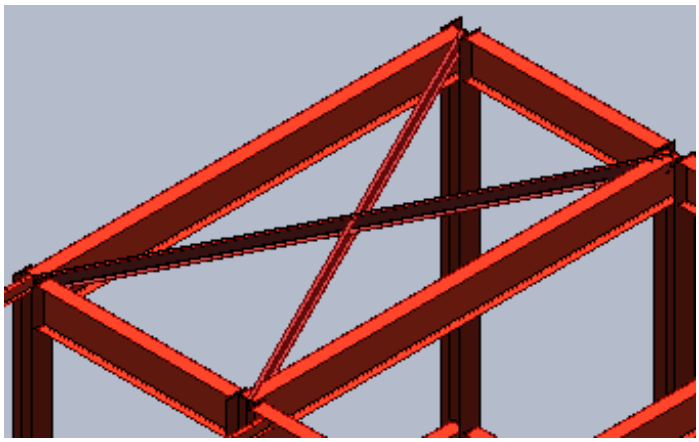


7. Select the second member to support the cross bracing. This member is the other "side" member to which the bracing connects.



8. Optionally, select one or two members to define the extent of the cross bracing. If you do not select these members, the software uses the end locations of the two selected members as the end locations of the cross bracing.

9. Click **Finish**.

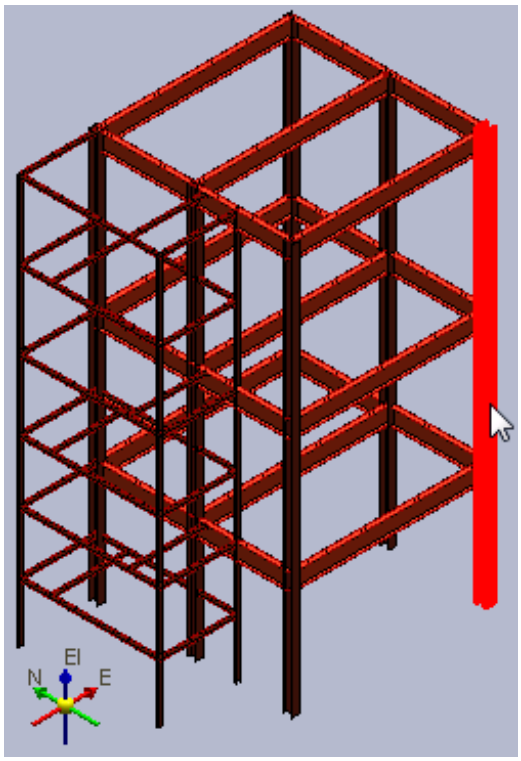


Place vertical chevron bracing

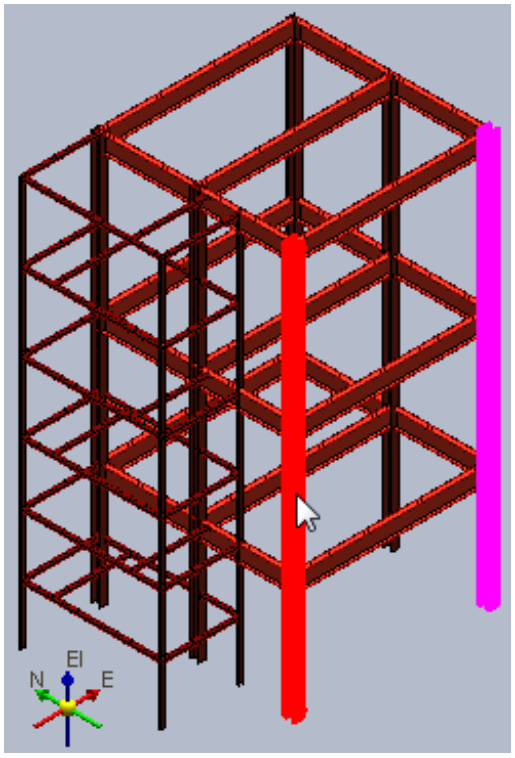
1. Click **Place X Bracing** on the vertical toolbar.
2. In the **Bracing Type** box, select **Chevron**.
3. In the **Type category** box, select the chevron bracing member type category to place.
4. In the **Type** box, select the chevron bracing member type to place.
5. Set the **Section Name**, **Cardinal Point**, and **Angle** options as required.

   TIP You can set the beta angle relative to your rotated coordinate system by activating **PinPoint** and selecting the rotated coordinated system as the active coordinate system in the **Coordinate System** box.

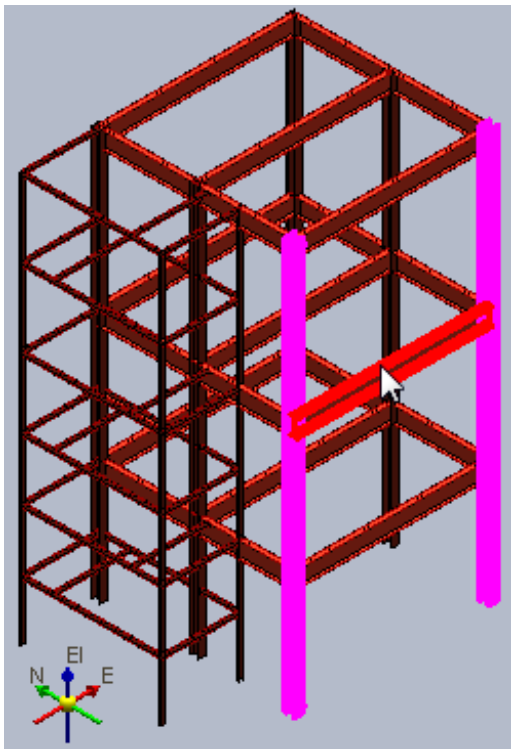
6. Select the first column to support the chevron bracing.



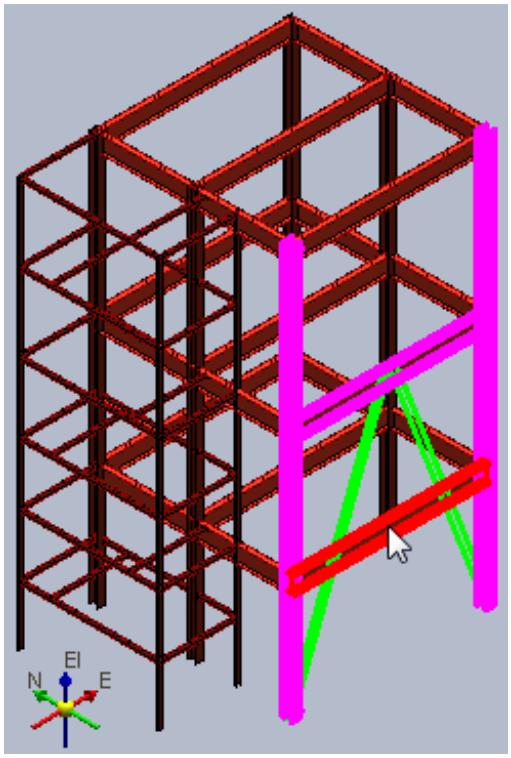
7. Select the second column to support the chevron bracing.



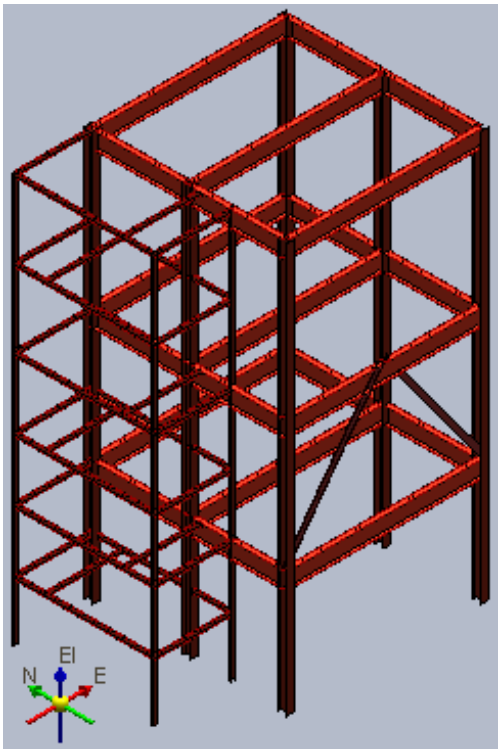
8. Select one beam to support the chevron bracing.



9. Optionally, select a second beam.



10. Click **Finish**.

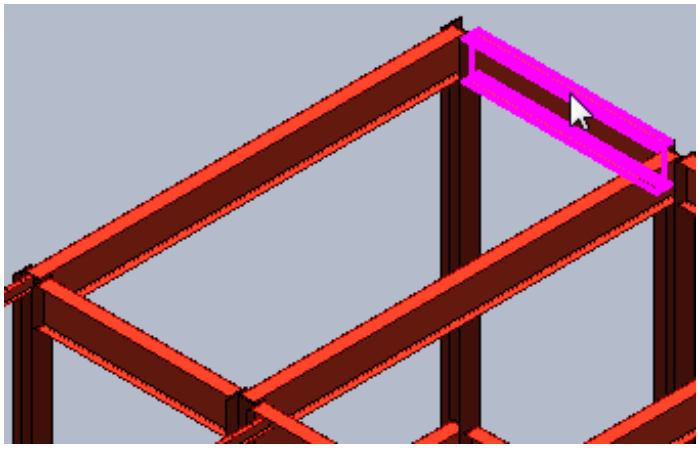


Place horizontal chevron bracing

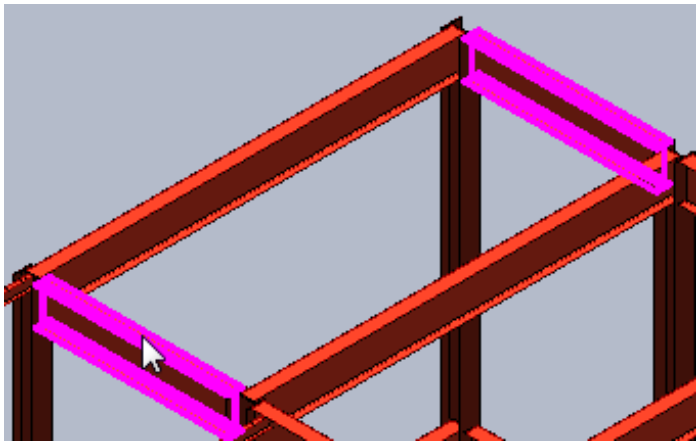
1. Click **Place X Bracing** on the vertical toolbar.
2. In the **Bracing Type** box, select **Chevron**.
3. In the **Type category** box, select the chevron bracing member type category to place.
4. In the **Type** box, select the chevron bracing member type to place.
5. Set the **Section Name**, **Cardinal Point**, and **Angle** options as required.

   TIP You can set the beta angle relative to your rotated coordinate system by activating **PinPoint** and selecting the rotated coordinated system as the active coordinate system in the **Coordinate System** box.

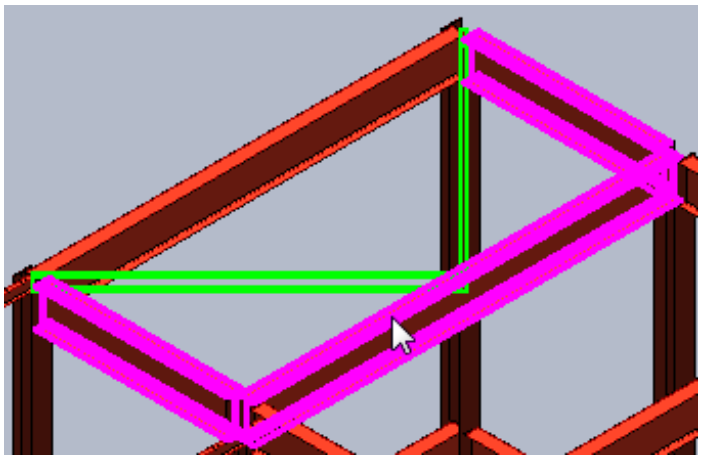
6. Select the first member to support the chevron bracing. This member is the first "side" member to which the bracing connects.



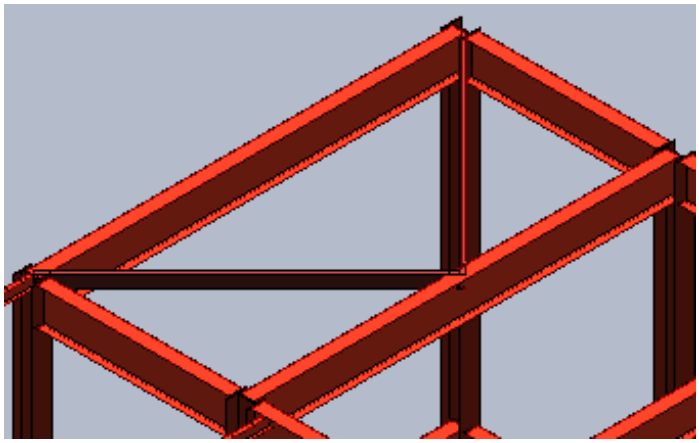
7. Select the second member to support the chevron bracing. This member is the second "side" member to which the bracing connects.



8. Select a member to support the chevron bracing. The mid-point of this member is used to define the location of the chevron bracing point.

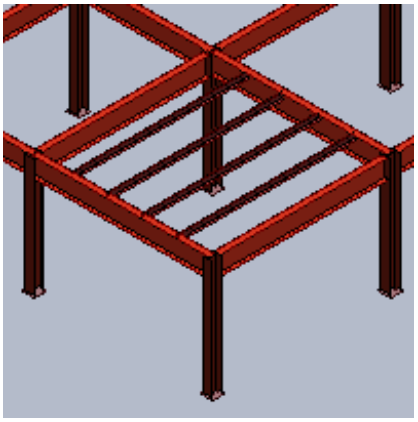


9. Optionally, select a second member. This member defines the end location for the wide part of the chevron bracing. If you do not select this optional member, the software uses the end location of the first and second members that you selected for the ending location of the chevron braces.
10. Click **Finish**.



SECTION 6

Place Framing Members



Places framing, secondary members, between two main supporting members typically to support a floor. The two supporting member must be coplanar but are not required to be of equal length or even the same member type thus providing lots of flexibility during placement. After placement, the framing members can be edited or deleted individually if needed.

For additional information about members, see *Members* (on page 22).

Place Framing Members Ribbon

Specifies the properties for the framing members that you are placing or editing.

Member Properties

Activates the **Member Properties** dialog box. You can use this dialog box to specify additional member properties, such as material and material grade, which you cannot set on the ribbon.

Select Supporting Member 1

Select the first supporting member for the framing members. If the two supporting members are different lengths, you will get better results if you select the shorter of the two members as supporting member 1.

Select Supporting Member 2

Select the second supporting member for the framing members.

Select Start Point

Select the location of the first framing member along the first supporting member. This option is available only when you set **Placement** to **Number and Spacing**.

Finish

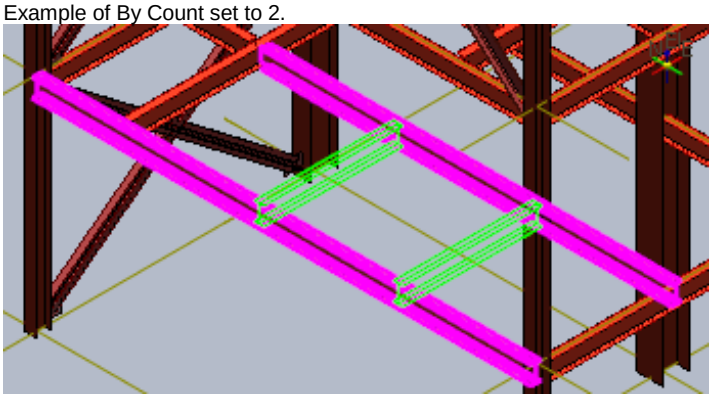
Click to place the framing members in the model.

Placement

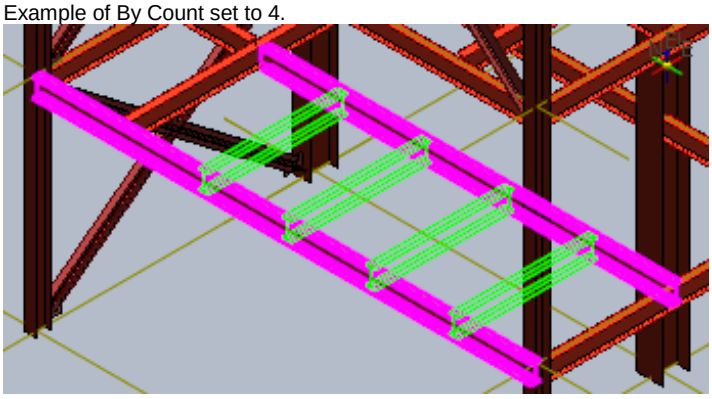
Select how to place the framing members between the two supporting members.

- **By Count** - Select this option if you want to specify the exact number of framing members to place. Type the number of framing members in the **Count** box. The spacing between the framing members is determined by the length of the supporting member divided by the number of framing members plus one.

Example of By Count set to 2.



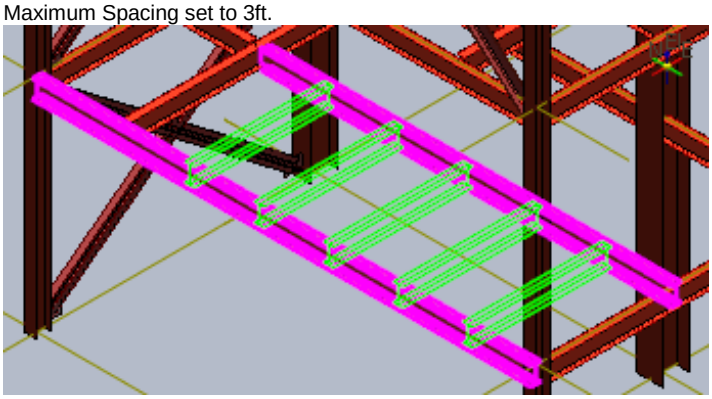
Example of By Count set to 4.



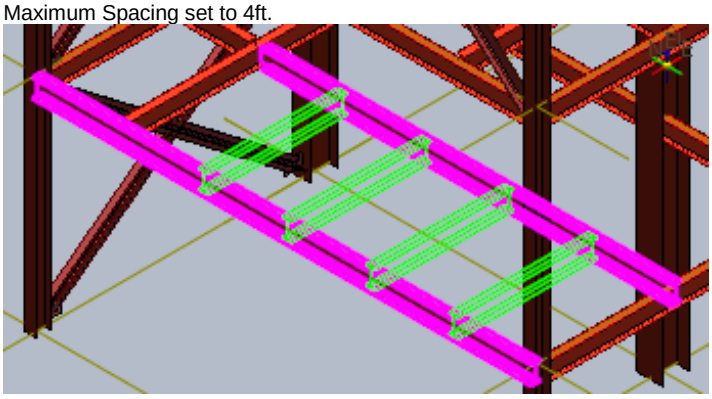
- **Equal Spacing** - Select this option to have the software determine the number of framing members to place based on the length of the supporting members and the value you specify in the **Maximum Spacing** box. The software places as many framing

members as needed so that the spacing between the framing members does not exceed the **Maximum Spacing** value. All framing members are equally spaced.

Maximum Spacing set to 3ft.



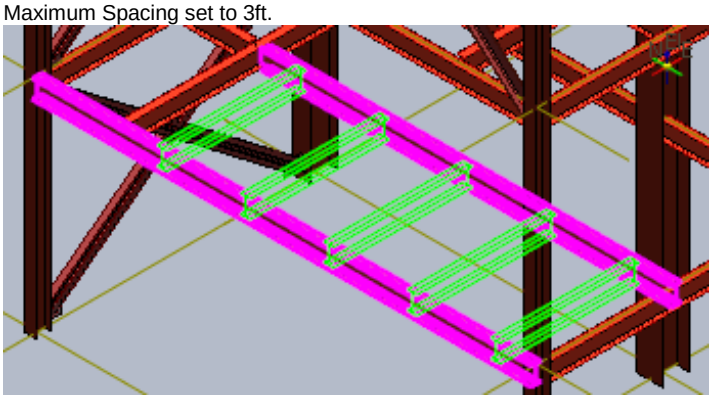
Maximum Spacing set to 4ft.



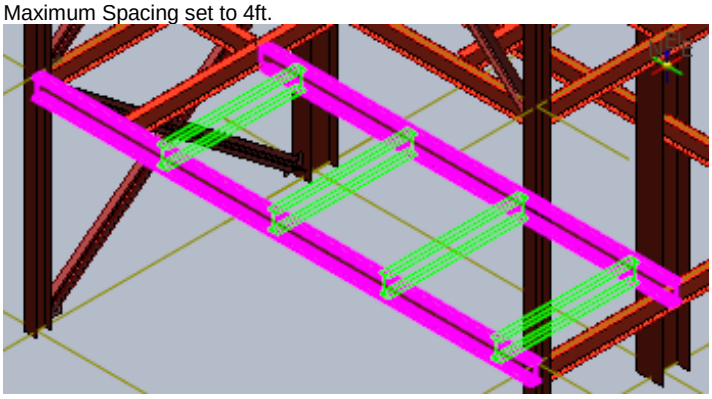
- **Best Fit** - Select this option to have the software determine the number of framing members to place based on the length of the supporting members and the value you specify in the **Maximum Spacing** box. The software places all framing members except for the first and the last with spacing equal to the **Maximum Spacing** value you

specified. The spacing for the first and last framing member is automatically determined by the software.

Maximum Spacing set to 3ft.



Maximum Spacing set to 4ft.



- **Number and Spacing** - Select this option to place the exact number of framing members that you specify in the **Count** box the same distance apart that you define in the **Maximum Spacing** box. In addition, you can optionally specify the location of the first framing member along the first selected supporting member using **Select Start Point** . The supporting member end to measure from is the nearest end when you selected the supporting member.

Maximum Spacing

Type the framing member spacing. If **Placement** is set to **Equal Spacing**, then this value is the maximum spacing that you will allow between the framing members. If **Placement** is set to **Best Fit** or **Number and Spacing**, then this value is used as the exact spacing between the interior framing members. This option is not available when **Placement** is set to **By Count**.

Count

Type the number of framing members to place. This option is available only when **Placement** is set to **By Count** or **Number and Spacing**.

Connection

Select the frame connection type to use for the member that you are placing. If you select **By Rule**, the software determines the frame connection to use based on the geometry between the member that you are placing and existing members in the model. If you select **More**, all available frame connections display from which you can select the frame

connection to use. For more information about frame connections, see *Frame Connections* (on page 24). This option is not available if you are editing an existing member. You can change the frame connection later by editing the frame connection directly.

System

Select the system to which the framing members belong. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see "Select System Dialog Box" in in *Place Linear Member Systems* (on page 35).

Type Category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

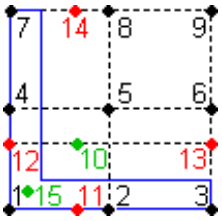
Type

Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Section Name

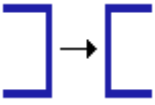
Defines the cross-section for the member. If you know the section name, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See Structure Reference Data Guide for more information about reference data.

Cardinal Point



Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Reflect



Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.

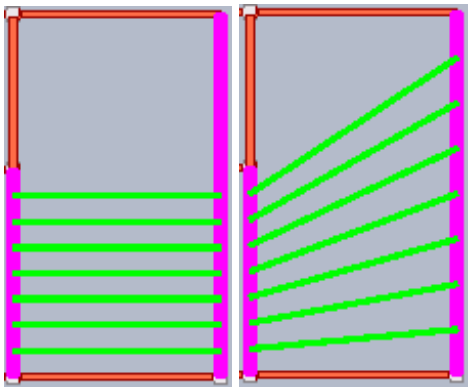
Offset

Specifies the framing member's offset from the supporting member's plane. This option is only available when **Connection** is set to a flush, seated, or centerline frame connection.

Position

Select how to position framing members if the two supporting members are not of equal

length or are not parallel. You can make the framing members perpendicular to the first supporting member that you selected, or to skew the framing members by the ratio difference of the two supporting members.



Perpendicular By Ratio

Side

Select to flip the framing members to the opposite side.

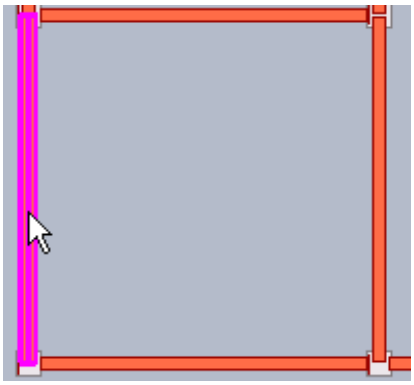
What do you want to do?

- *Position framing members using best fit* (on page 132)
- *Position framing members using count method* (on page 133)
- *Position framing members at equal spacing* (on page 135)
- *Position framing members using number and spacing method* (on page 136)
- *Copy framing members* (on page 136)

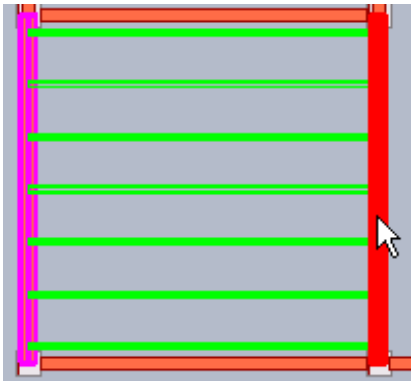
Position framing members using best fit

1. Click **Place Framing Members** on the vertical toolbar.
2. In the **Placement** box, select **Best Fit**.
3. In the **Max Spacing** box, type the distance to separate the interior framing members. The distance for the first and last framing members is determined by the software.
4. In the **Type category** box, select the member type category for the framing member being placed.
5. In the **Type** box, select the member type for the framing member being placed.
6. Set the **Section Name**.

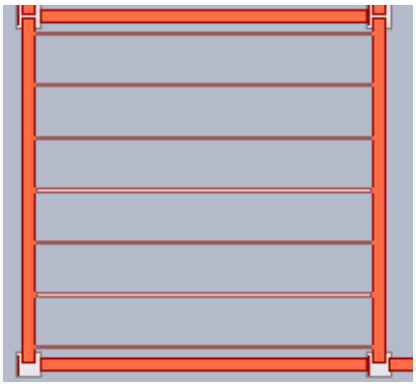
7. Select the first supporting member.



8. Select the second supporting member.



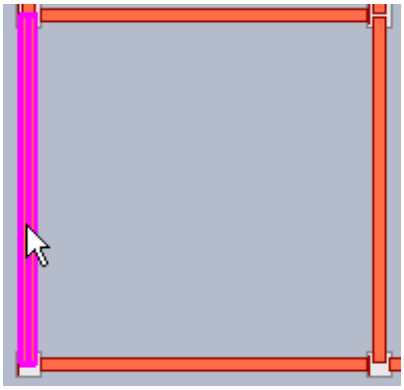
9. Set the **Reflect**, **Cardinal Point**, and **Offset** options as required.
10. Click **Finish**.



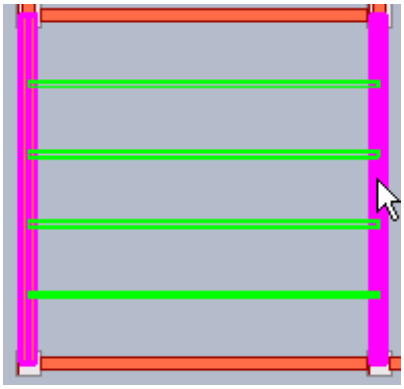
Position framing members using count method

1. Click **Place Framing Members** on the vertical toolbar.
2. In the **Placement** box, select **By Count**.
3. In the **Count** box, type the number of framing members to place.

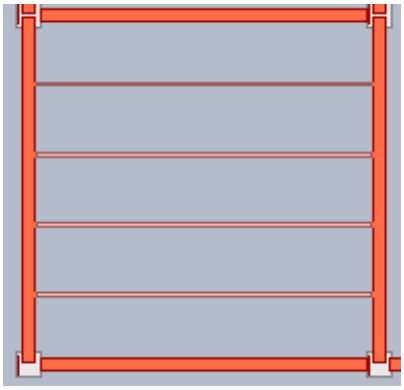
4. In the **Type category** box, select the member type category for the framing member being placed.
5. In the **Type** box, select the member type for the framing member being placed.
6. Set the **Section Name**.
7. Select the first supporting member.



8. Select the second supporting member.

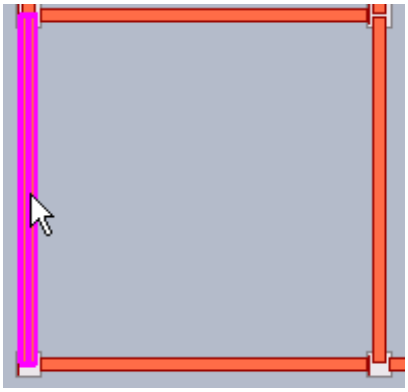


9. Set **Reflect**, **Cardinal Point**, and **Offset** as required.
10. Click **Finish**.

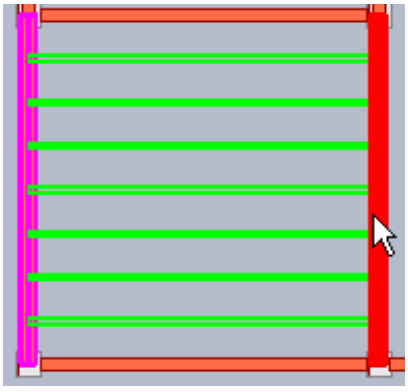


Position framing members at equal spacing

1. Click **Place Framing Members** on the vertical toolbar.
2. In the **Placement** box, select **Equal Spacing**.
3. In the **Max Spacing** box, type the maximum distance that you will allow between the framing members. The distance between the framing members will be equal to or less than this distance.
4. In the **Type category** box, select the member type category for the framing member being placed.
5. In the **Type** box, select the member type for the framing member being placed.
6. Set the **Section Name**.
7. Select the first supporting member.

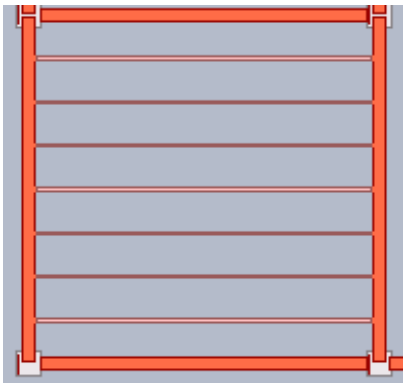


8. Select the second supporting member.



9. Set **Reflect**, **Cardinal Point**, and **Offset** options as required.

10. Click **Finish**.



Position framing members using number and spacing method

1. Click **Place Framing Members** on the vertical toolbar.
2. In the **Placement** box, select **Number and Spacing**.
3. In the **Count** box, type the number of framing members to place.
4. In the **Maximum Spacing** box, type the distance between the framing members.
5. In the **Type category** box, select the member type category for the framing member being placed.
6. In the **Type** box, select the member type for the framing member being placed.
7. Set the **Section Name**, **Cardinal Point**, and **Offset** options as required.
8. Select the first supporting member. If you are going to define a starting location for the framing members, select the first supporting member near the end from which you want to define the framing member's starting location.
9. Select the second supporting member.
10. Optionally, select the starting location of the first framing member along the first supporting member.
11. Click **Finish**.

Copy framing members

1. Click **Select** on the vertical toolbar.
2. Select **Member Systems** in the locate filter.
3. Select the framing members to copy. Press CTRL as you select the members.
4. Press CTRL+C or click **Edit > Copy** to copy the members to the clipboard.
5. Select a reference point (a from-point) for the members being copied.
6. Press CTRL+V or click **Edit > Paste** to paste the members from the clipboard.

*The **Paste** dialog displays.*

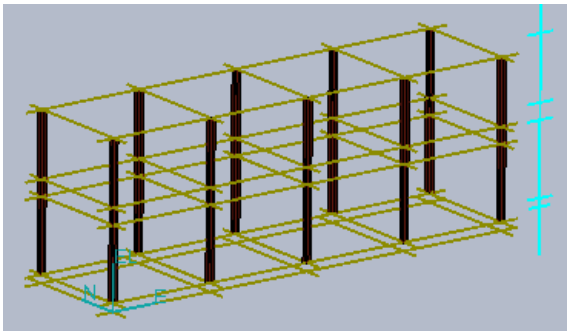
7. In the **Paste** dialog box, select new systems and dependencies for the paste members. If you do not select new dependencies for the pasted members in the **Paste** dialog box, the pasted members are placed in the same location as the copied members.
8. Click **OK** on the **Paste Special** dialog box.

NOTE Assembly Connections are not copied to the pasted members.

SECTION 7

Place Columns at Grid Intersections

Places columns between two selected elevations at selected grid intersections. You must have two elevation planes and one grid plane defined before you can use this command. Use the **Grids** task to create the planes. After placement, the columns can be edited or deleted individually if needed. For additional information about members, see *Members* (on page 22).



Place Columns at Grid Intersections Ribbon

Specifies the properties for the columns that you are placing or editing.

Member Properties

Activates the **Member Properties** dialog box. You can use this dialog box to specify additional member properties, such as material and material grade, which you cannot set on the ribbon.

Select Elevation Plane 1

Specify the bottom elevation for the columns.

Select Elevation Plane 2

Specify the top elevation for the columns.

Select Grid Intersections

Drag a fence around all the grid intersections where you want columns placed. You can drag multiple fences if needed.

Finish

Click to place the columns in the model.

Cancel

Clears the selected grid intersections.

System

Select the system to which the columns belong. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see "Select System Dialog Box".

Type Category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

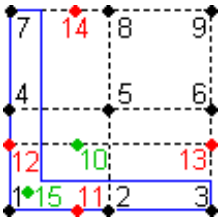
Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Section Name

Defines the cross-section for the member. If you know the section name, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See Structure Reference Data Guide for more information about reference data.

Cardinal Point

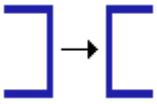
Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.



Angle

Defines the angle, in degrees or radians, by which the cross-section is rotated about the member axis. The zero-degree position is either the Z-axis or the X-axis of the active coordinate system depending on the member orientation. The active coordinate system is set using the **Coordinate System** control in **PinPoint**  . The **Angle** property is not used when placing members using the **Seated**, **Flush**, or **Centerline** frame connections.

Reflect

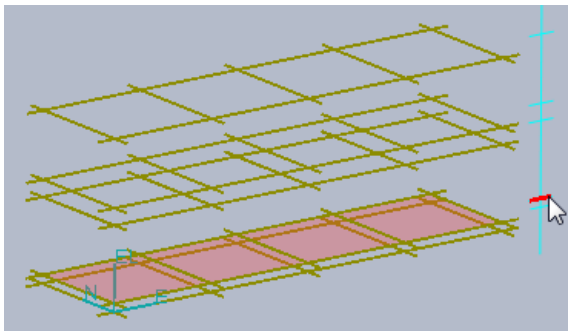


Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.

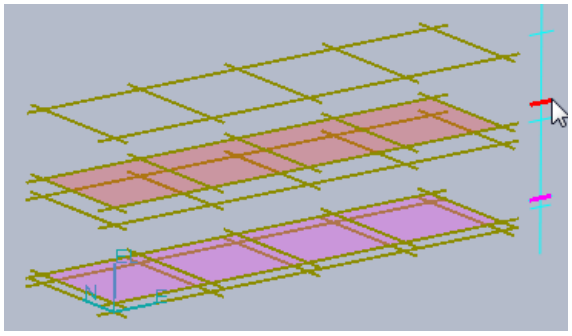
Place columns at grid intersections

1. Click **Place Columns at Grid Intersections** on the vertical toolbar.
2. In the **Type category** box, select the member type category to place.
3. In the **Type** box, select the member type to place.
4. Set the **Section Name**, **Cardinal Point**, and **Angle** options as required.

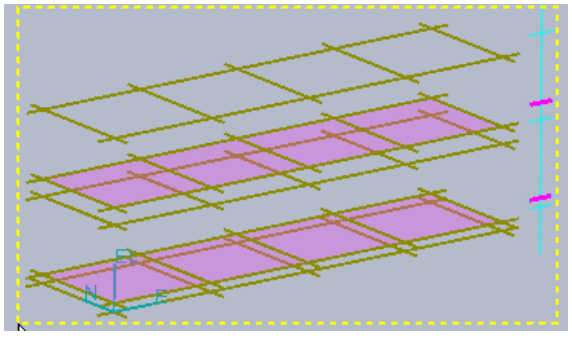
   TIP You can set the beta angle relative to your rotated coordinate system by activating **PinPoint** and selecting the rotated coordinated system as the active coordinate system in the **Coordinate System** box.
5. Select the bottom elevation plane for the columns.



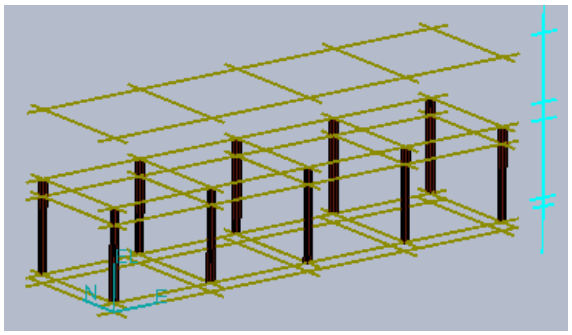
6. Select the top elevation plane for the columns.



7. Drag a fence around the grid intersections where you want columns.

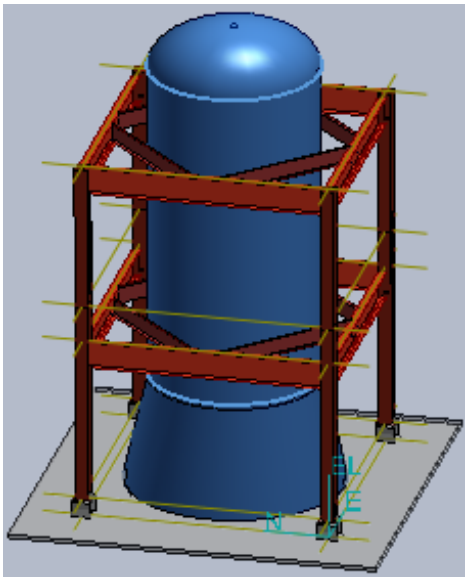


8. Click **Finish**.



SECTION 8

Place Vessel Supports



Places structural members that support vertical vessels directly (typically four lugs that rest on the members) or that support the grating around the vessel. This command considers the vessel diameter and clearance, or the bolt circle diameter (lug-hole to lug-hole diameter) to determine member positions. Based on the cross-section type chosen (I-section or channel), the command also considers the gage (the distance to the bolt hole location) in positioning the member. Because the equipment is often not centered within the main framing of the opening, the command accommodates such conditions and displays appropriate messages if the framing is not feasible. In general, the four supporting members must form a square.

This command is currently limited to configurations where the lugs are positioned North, South, East, and West, or at 45 degrees to these axes. This command only supports the placement of W, M, HP, S, C, MC, BUI, BUC, BUTee, BUIUE, and BUBoxFM sections.

For additional information about members, see *Members* (on page 22).

Place Vessel Supports Ribbon

Specifies the properties for the vessel support members that you are placing or editing.

Select Vessel

Select the vessel around which to place the support members.

Select Members

Select the primary supporting members from which the vessel support members are placed.

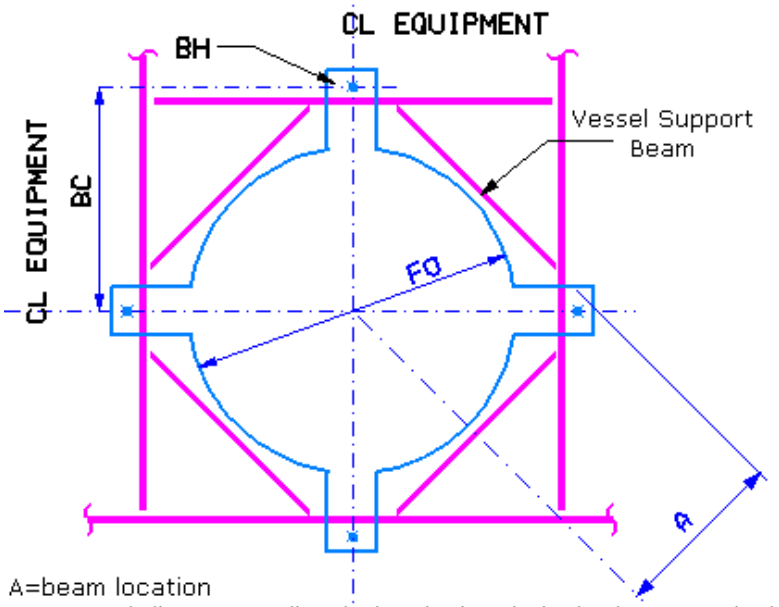
Finish

Click to place the vessel support members in the model.

Lug Orientation

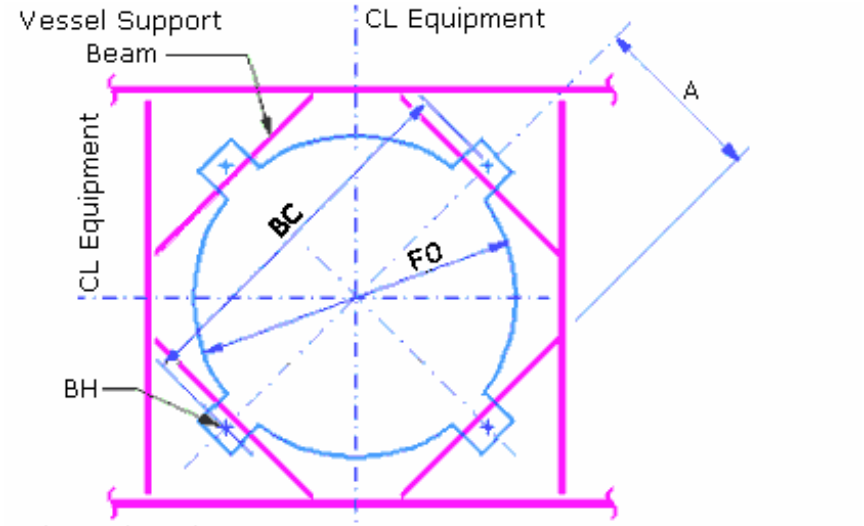
Select the orientation of the vessel lugs relative to the coordinate system north.

Select **0 Degrees** if the vessel lugs are located like those in the figure below:



A=beam location
FO=vessel diameter + (insulation * 2) + (2 inch clearance * 2)
BC=bolt circle
BH=bolt hole diameter

Select **45 Degrees** if the vessel lugs are located like those in the figure below:



A=beam location
FO=vessel diameter + (insulation * 2) + (2 inch clearance * 2)
BC=bolt circle
BH=bolt hole diameter

Easting

Specifies the easting coordinate of the vessel center. If you select a vessel, the software automatically finds this value for you. If you are placing supports before the vessel is modeled, type the easting coordinate of the vessel in this box. This coordinate is relative to the active coordinate system defined in the **PinPoint** ribbon.

Northing

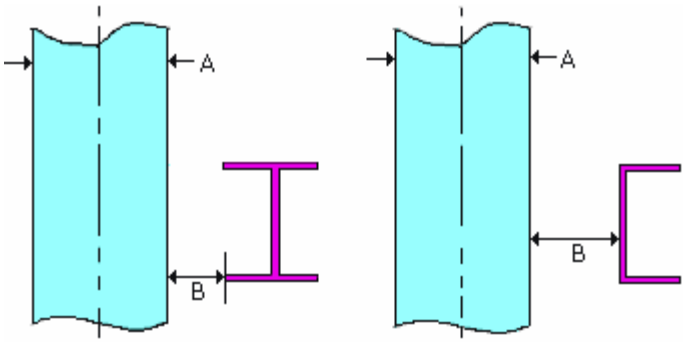
Specifies the northing coordinate of the vessel center. If you select a vessel, the software automatically finds this value for you. If you are placing supports before the vessel is modeled, type the northing coordinate of the vessel in this box. This coordinate is relative to the active coordinate system defined in the **PinPoint** ribbon.

Vessel Diameter

Specifies the vessel diameter. If you select a vessel, the software automatically enters the diameter for you. If you are placing supports before the vessel is modeled, type the vessel diameter in this box. This option is only available when **Lug Orientation** is set to **0 Degrees**.

Clearance

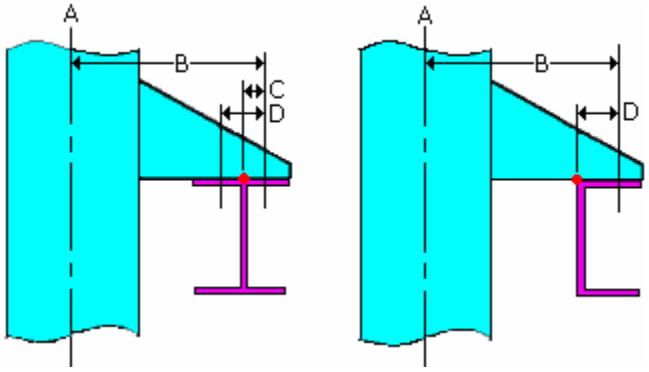
Specifies the distance between the vessel support members and the vessel itself. This option is only available when **Lug Orientation** is set to **0 Degrees**.



A=Vessel Diameter
B=Clearance

Bolt Circle

Specifies the distance between the bolts on the opposite sides of the vessel. This value is used to calculate the member line location (red dot in figure below, figure shows cardinal point 8 being used for the I-beam and cardinal point 7 for the channel). This option is only available when **Lug Orientation** is set to **45 Degrees**.



A=Vessel Centerline
B=Bolt Circle divided by 2
C=Bolt Gage divided by 2
D=Bolt Gage

System

Select the system to which the vessel support members belong. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see Select System Dialog Box.

Type Category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

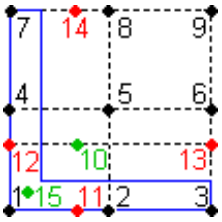
Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Section Name

Defines the cross-section for the member. If you know the section name, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See Structure Reference Data Guide for more information about reference data. This command only supports the placement of W, M, HP, S, C, MC, BUI, BUC, BUTee, BUIUE, and BUBoxFM sections.

Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

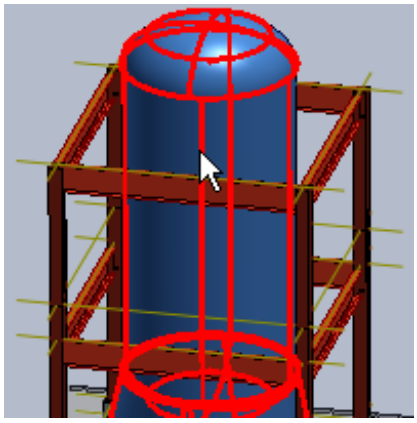


What do you want to do?

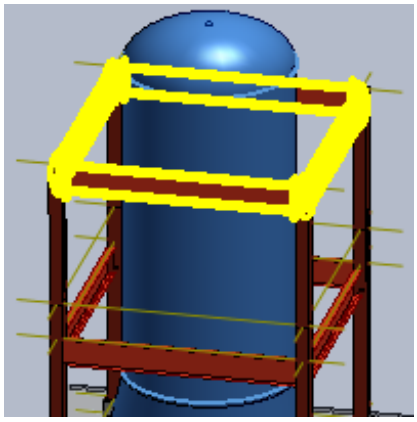
- *Support vessels with 0 degree oriented lugs* (on page 146)
- *Support vessels with 45 degree oriented lugs* (on page 147)

Support vessels with 0 degree oriented lugs

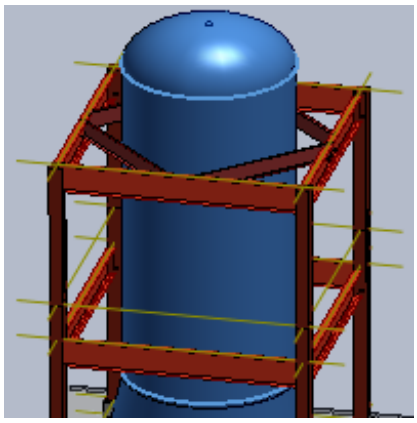
1. Click **Place Vessel Supports** on the vertical toolbar.
2. Select the vessel around which to place the supports.
   -OR-
   Key in the Easting and Northing coordinates of the vessel's center.



3. Set **Lug Orientation** to **0 Degrees**.
4. Key in the clearance between the side of the vessel and the support members being placed.
5. Select a system, type, section, and cardinal point for the support members. This command only supports the placement of W, M, HP, S, C, MC, BUI, BUC, BUTee, BUIUE, and BUBoxFM sections.
6. Select the four main support members in clockwise order.



7. Click **Finish**.



Support vessels with 45 degree oriented lugs

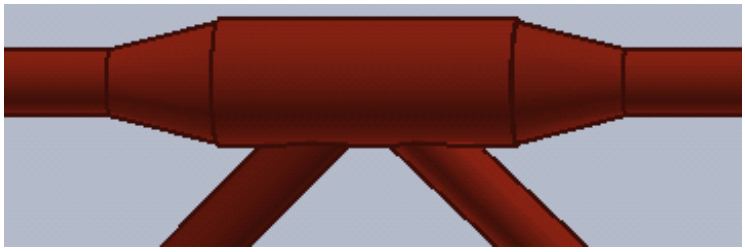
1. Click **Place Vessel Supports** on the vertical toolbar.
2. Select the vessel around which to place the supports.
   -OR-
   Key in the Easting and Northing coordinates of the vessel's center.
3. Set **Lug Orientation** to **45 Degrees**.
4. Key in the bolt circle diameter.
5. Select a system, type, section, and cardinal point for the support members. This command only supports the placement of W, M, HP, S, C, MC, BUI, BUC, BUTee, BUIUE, and BUBoxFM sections.
6. Select the four main support members in clockwise order.
7. Click **Finish**.

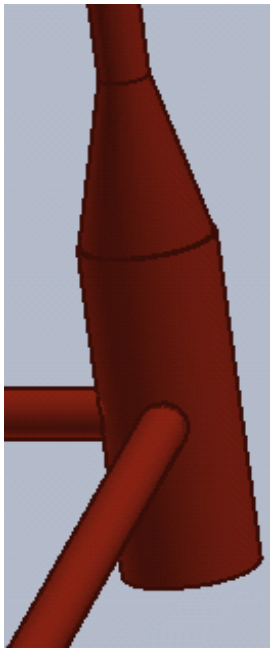
SECTION 9

Place Can

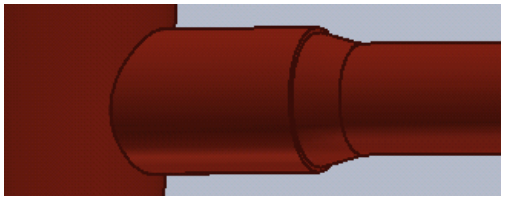
Places a can in the model. A can is a reinforcing connection piece on a member to increase the surface area available for connections or to provide increased wall thickness or both. You can place cans anywhere along a linear member (the support leg) including the end of the member. There are three can types:

- Inline can - This can is placed anywhere along the length of a support leg. The support leg is split into two parts and the can is placed between those parts.





- End can - This can is placed at the end of the support leg. As a general rule, the end of the support leg does not frame into another member.
- Stub-end can - This can is also placed at the end of the support leg, which generally does frame into another member.



The software calculates the can length based on any transverse members that you select and the L2 and L3 parameters that you define.

Can Limitations

- SmartSketch precision placement commands such as **Point Along** and **PinPoint** are not available when placing an in-line can. The in-line can's location is determined by where you select the supporting member or by the location of any transverse members assigned to the can.
- Stub-in and end cans locations are determined by the frame or split connection that you select during placement.
- The linear member that you select to be the supporting leg must be a designed member with a BUTube cross-section. You cannot select curved members for the supporting leg or for the transverse members.

Place Can Ribbon

Can Properties

Activates the **Can Properties** dialog box. Use this dialog box to specify additional properties, such as material and material grade, which you cannot set on the ribbon. For more information, see *Can Properties Dialog Box* (on page 159).

Select Primary Member

Select the support leg on which to place the can. The support leg is split at the location you specify and highlights in red during can placement. If you are placing an inline can, you can select a BUTube designed member. You cannot select a curved member as the primary member. If you are placing an end can, you must select an unsupported frame connection. If you are placing a stub-end can, you can select any split connection or frame connection except unsupported frame connections.

If you are modifying an existing can, this option is not available.

Select Member Part

Select the member part on which to place the stub-end can. This option is only available with the **Can Type** is set to a stub-end can and you have selected a split connection for the can's location.

Select Transverse Members

Select the transverse members with which you want to base the can length. All transverse members that are related to the can highlight in blue. You can select any member systems, except curved members, that are connected to the primary member. You can use **Cancel** ✕ to clear selected transverse members and **Accept** ✓ to confirm the transverse members that you selected are the ones you want to use.

If you are modifying more than one existing can, this option is not available.

Finish

Click to place the can in the model.

✕ Cancel

Rejects the transverse members that you have selected.

✓ Accept

Confirms the transverse members that you have selected.

Primary Member Information

Activates a small dialog box that displays the name, outside diameter, inside diameter, and plate thickness of the selected primary member. This information is useful when selecting an appropriate can type from the Catalog or when defining custom can specifications.

If you are modifying an existing can, this dialog box displays the name, outside diameter, inside diameter, and plate thickness of the support legs on either side of the selected can.

Can Type

Specifies the type of can to place: stub-end, inline, or end can. When you select a can type, the defaults settings from the Catalog are loaded.

Tube / Cone Parameters

Click to swap between defining cone parameters and defining tube parameters.

Tube Parameters

Outer/Inner Diameter

Specifies the can outer diameter or the can inner diameter depending on the can type you have selected.

- If you selected a pre-defined can from the Catalog, the diameter value from the Catalog is displayed.
- If you selected a Custom Can-OD Match from the Catalog, the outer diameter of the selected primary member is displayed.
- If you selected a Custom Can-ID Match from the Catalog, the inner diameter of the selected primary member is displayed.
- If you selected a Custom Can-User Defined from the Catalog, you must type a diameter value in this box.

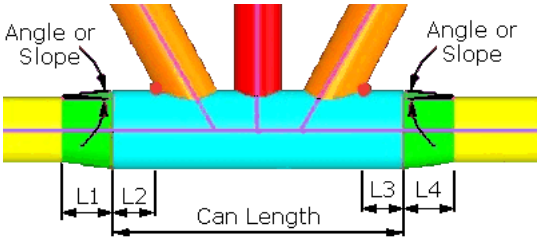
Can Thickness

Select the plate thickness for the can plate.

Computation Method

Specifies how you want Smart 3D to measure the L2 and L3 minimum distances.

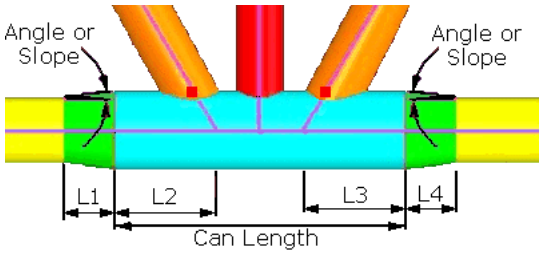
- If you select **Hull-Factor**, the software calculates the outside diameter of the can times the value you type in the **Factor** box. The result is used as the minimum L2 and L3 distances which are measured from the corresponding end of the can to the point on the transverse member hull closest to that end of the can.
- If you select **Hull-Length**, the software uses the distance you specify in the **Length** box for L2 and L3, which are measured from the corresponding end of the can to the point on the transverse member hull closest to that end of the can.



L2 and L3 Lengths when using a Hull Extension Method

- If you select **Centerline-Factor**, the software calculates the outside diameter of the can times the value you type in the **Factor** box. The result is used as the minimum L2 and L3 distances, which are measured from the corresponding end of the can to the centerline of the transverse member closest to that end of the can.

- If you select **Centerline-Length**, the software uses the distance you specify in the **Length** box for L2 and L3, which are measured from the corresponding end of the can to the centerline of the transverse member closest to that end of can.



L2 and L3 Lengths when using a Centerline Method

Factor

Specifies the factor to use in the extension method calculation. The default value is 0.5.

Length

Specifies the absolute distance to use in the extension method calculation. If you type a length that is shorter than the American Petroleum Institute's minimum allowable length, a message displays that one or more specified parameters are invalid.

Cone Parameters

Cones By

Specifies how you want to define the length of the cone transitions.

- If you select **Angle** the length of the transition is determined by the geometry of the can diameter, the support member cross-section, and the angle specified in the **Cone Angle** box. This is shown as **Angle** in the figures above.
- If you select **Length** the length of the transition is defined explicitly in the **Cone Length** box. This distance is shown as L1 and L4 in the figures above.
- If you select **Slope**, the length of the transition is determined by the geometry of the can diameter, the support member cross-section size, and the slope value specified in the **Cone Slope** box (the denominator of the slope). This is shown as **Slope** in the figures above.

NOTE During placement, both cone transitions are placed using one set of values. If you want the cone transitions on either end of the can to have different lengths or different angles, you can change these values independently on the *Can Properties Dialog Box* (on page 159) after placement.

Cone Angle

Specifies the angle of the cone transition. The length of the cone is adjusted so that the entered angle is obtained. The software does not create a cone if the angle you specify is less than 1 degree or greater than 89.0 degrees.

Cone Length

Specifies the length of the cone transition from the end of the support member to the start of the can. This distance is shown as **L1** and **L4** in the figures above.

Cone Slope

Specifies the slope of the cone transition. The length of the cone is adjusted so that the entered slope is obtained. The software does not create a cone if the slope you specify is greater than 100 (1/100) or less than 1^{-6}.

Cone Thickness

Select the plate thickness for the cone.

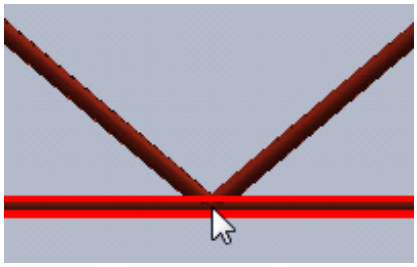
What do you want to do?

- *Place an in-line can* (on page 152)
- *Place an in-line can on cross bracing* (on page 153)
- *Place an end can* (on page 155)
- *Place a stub-end can* (on page 156)
- *Add a transverse member to a can* (on page 157)
- *Remove a transverse member from a can* (on page 158)

Place an in-line can

1. Click **Place Can** on the vertical ribbon.
2. From the Catalog browser or from **Can Type** on the ribbon, select the in-line can rule to place from the Catalog.

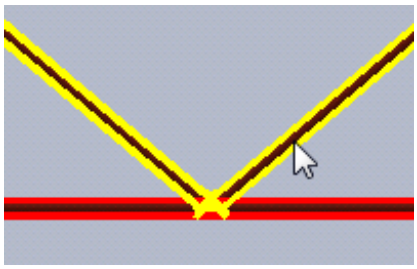
   TIP Select one of the Custom Inline Can types to change individual can parameters.
3. Select the location of the in-line can along the supporting leg member.



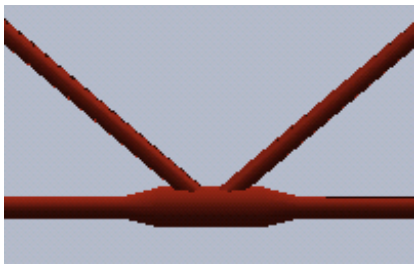
4. Click **Primary Member Information** on the ribbon.

   The software displays the name, outside diameter, inside diameter, and plate thickness of the selected supporting leg.

5. Select the transverse members that participate in the can. Transverse members are used in calculating the can length.

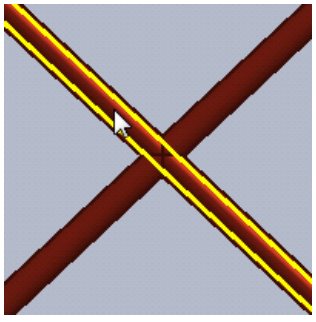


6. Click **Accept** ✓.
7. If you are placing a Custom Inline Can type, change the diameter, thickness, computation method, factor, and cone parameters on the ribbon to meet your needs. If you are placing a predefined can from the Catalog, these settings are read-only.
8. Click **Finish**.



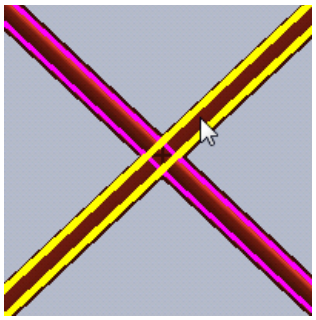
Place an in-line can on cross bracing

1. Click **Place Split** on the vertical toolbar.
2. In the **Place** box, select **By Object**.
3. In the **Split Status** box, select **Split First**.
4. Select the cross brace to split.



5. Click **Accept** ✓.

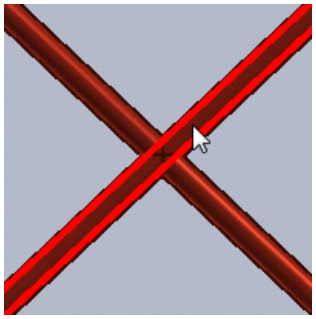
6. Select the other cross brace.



7. Click **Finish**.
8. Click **Place Can** on the vertical ribbon.
9. From the Catalog browser or from **Can Type** on the ribbon, select the in-line can rule to place from the Catalog.

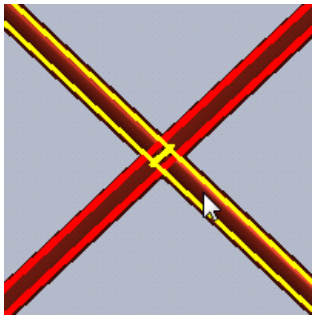
   TIP Select one of the Custom Inline Can types to change individual can parameters.

10. Select the cross brace you selected in step 6 (the cross brace that you did not split) near the intersection of the two cross braces.



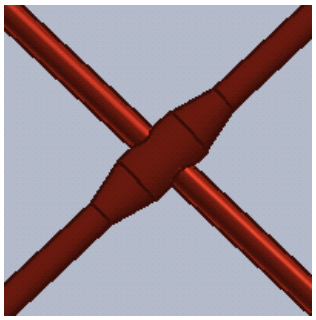
   The software prompts you to select the transverse members.

11. Select the cross brace that you split (the cross brace you selected in step 4).



12. Click **Accept** ✓.
13. If you are placing a Custom Inline Can type, change the diameter, thickness, computation method, factor, and cone parameters on the ribbon to meet your needs. If you are placing a predefined can from the Catalog, these settings are read-only.

14. Click **Finish**.

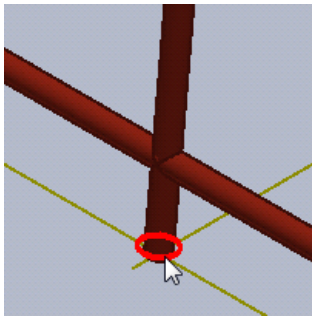


Place an end can

1. Click **Place Can** on the vertical ribbon.
2. From the Catalog browser or from **Can Type** on the ribbon, select the end can rule to place from the Catalog.

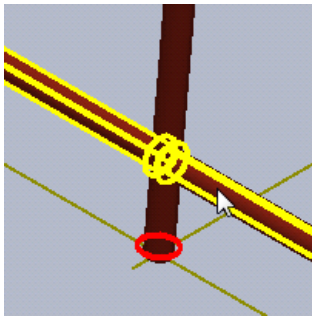
   TIP Select one of the Custom End Can types to change individual can parameters.
3. Select the unsupported frame connection at the member end where you want the end can.

   TIP End cans are only placed on unsupported frame connections.

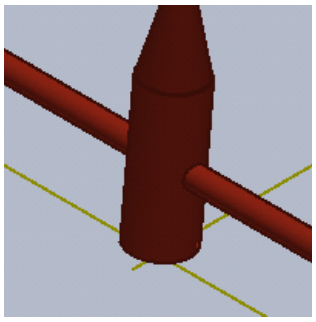


4. Click **Primary Member Information** on the ribbon.

   The software displays the name, outside diameter, inside diameter, and plate thickness of the selected supporting member.
5. Select the transverse members, if any, that participate in the can. Transverse members are used in calculating the length of the can.



6. Click **Accept** ✓.
7. If you are placing a Custom End Can type, change the diameter, thickness, computation method, factor, and cone settings on the ribbon to meet your needs. If you are placing a predefined can from the Catalog, these settings are read-only.
8. Click **Finish**.



Place a stub-end can

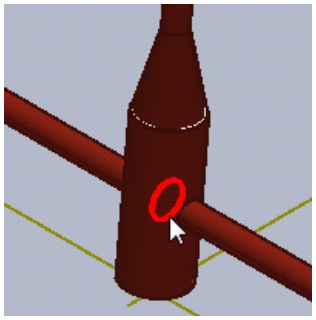
1. Click **Place Can** on the vertical ribbon.
2. From the Catalog browser or from **Can Type** on the ribbon, select the stub-end can rule to place from the Catalog.

   TIP Select one of the Custom Stub End Can types to change individual can parameters.
3. Select the frame connection at the member end where you want the stub-end can.

   TIP Stub-end cans cannot be placed on unsupported frame connections.

   or

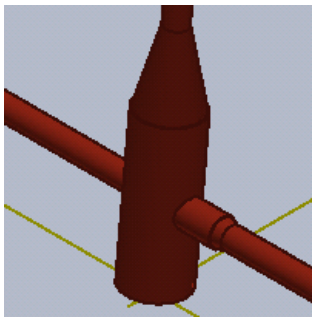
   Select the split connection where you want to place the stub-end can.



4. If placing the stub-end can at a split connection, select the member part on which to place the stub-end can.
5. Click **Primary Member Information** on the ribbon.

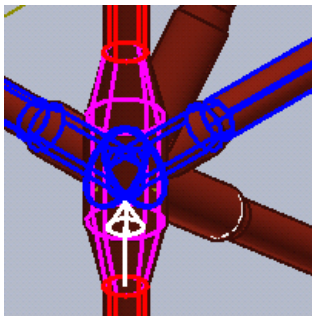
   The software displays the name, outside diameter, inside diameter, and plate thickness of the selected supporting member.
6. Select the transverse members, if any, that participate in the can. Transverse members are used in calculating the can length.

7. Click **Accept** ✓.
8. If you are placing a Custom Stub End Can type, change the diameter, thickness, computation method, factor, and cone settings on the ribbon to meet your needs. If you are placing a predefined can from the Catalog, these settings are read-only.
9. Click **Finish**.

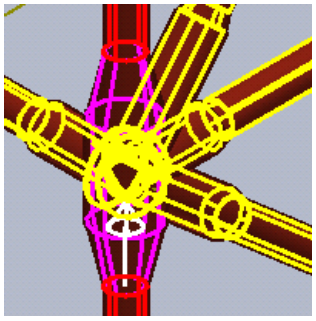


Add a transverse member to a can

1. Click **Select** on the vertical toolbar.
2. Select the can to contain the new the transverse member.

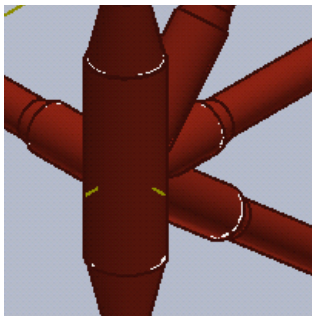


3. Select **Transverse Members** on the ribbon.
4. Select the member to add to the can.



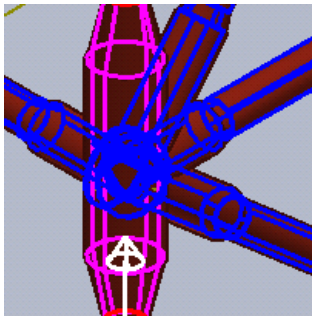
5. Click **Accept** ✓.

6. Click **Finish**.

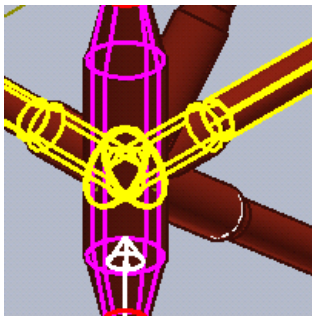


Remove a transverse member from a can

1. Click **Select** on the vertical toolbar.
2. Select the can containing the transverse member you want to remove.

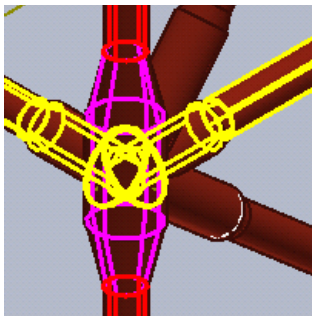


3. Select **Transverse Members** on the ribbon.
4. Select the member to remove from the can.



5. Click **Accept** ✓.

6. Click **Finish**.



Can Properties Dialog Box

Specifies the properties for the can that you are editing.

See Also

Occurrence Tab (Can Properties Dialog Box) (on page 159)
Can Rule Tab (Can Properties Dialog Box) (on page 164)
Cross Section Tab (Can Properties Dialog Box) (on page 170)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

Occurrence Tab (Can Properties Dialog Box)

Specifies the properties for the select can.

Category

Select the properties that you want to view for the member. Currently, only Standard properties are available for BUCan designed member.

Preview

Displays a graphic indicating the different dimensions that can be specified for the selected designed member.

Standard

Name

Displays the name of the can. The can name is based on the **Name Rule** selection. If you want to type a new name for the can, in the **Name Rule** box, select **User Defined**, and then type a name for the can in the **Name** box.

Name Rule

Specify the naming rule to use to name this can. You can select one of the listed rules or select **User Defined** to specify the can name yourself in the **Name** box.

- **Default Rule** - Names the can using the syntax: BUCan<*Can Type*>-<*site ID*>-<*Index*> where: <Can Type> is the can type selected from the Catalog, <site ID> is the location ID defined in Project Management, and <Index> is a 4-digit unique index number that starts at 0001.

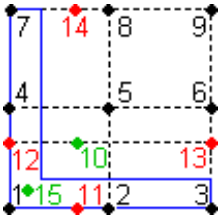
Parent System

Specifies the name of the parent system. You can define new systems in the Systems and Specifications task.

Section Standard

Specifies the section library from which you selected the section name. Sections are defined in the reference data.

Cardinal Point

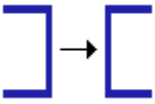


Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

Defines the angle by which the section is rotated about the member axis.

Reflect



Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.

Type category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Priority

Select the priority to assign to the designed member. The priority is used to group members.

Centroid X

Displays the location of the centroid along the local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Centroid Y

Displays the location of the centroid along the local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about X (Ixx)

Displays the moment of inertia for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about Y (Iyy)

Displays the moment of inertia for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Statical Moment (Sw)

Displays the warping statical moment. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about X (Sxx)

Displays the section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about Y (Syy)

Displays the section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Torsional Moment of Inertia (J)

Displays the torsional moment of inertia for the section. This property is read- only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Constant (Cw)

Displays the warping constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Flexural Constant (H)

Displays the flexural constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Polar Radius of Gyration about Shear Center (ro)

Displays the polar radius of gyration about the shear center. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about X axis (Rxx)

Displays the radius of gyration for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Y axis (Ryy)

Displays the radius of gyration for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Principle XY (Rxy)

Displays the radius of gyration about the principle xy-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about X (Zxx)

Displays the plastic section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about Y (Zyy)

Displays the plastic section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Is Hollow

Displays **True** if the cross-section is hollow, such as a tube or can. Displays **False** if the cross-section is solid, such as a bar, or open, such as an I-beam.

Is Symmetric About X

Displays **True** if the cross-section is symmetric about the local x-axis of the cross-section.

Is Symmetric About Y

Displays **True** if the cross-section is symmetric about the local y-axis of the cross-section.

Diameter - Start

Displays the diameter of the start cone.

Diameter - End

Displays the diameter of the end cone.

Length - Start Cone

Displays the length of the start cone, which is the cone closest to the start of the supporting member.

Length - End Cone

Displays the length of the end cone.

Section Properties

Displays how section properties are defined.

Area

Displays the cross-section area for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Depth

Displays the depth for the section. This property is read-only. To edit this value, you need to edit the cross- section's properties in the Catalog task or in the corresponding workbook.

Width

Displays the flange width for the section. This property is read-only. To edit this value, you

need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Perimeter

Displays the outside perimeter distance for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Unit Weight

Displays the unit weight.

Length Extension

Displays the length extension, which is the minimum distance between the transverse member and the end of the tube part of the can.

Can Length

Displays the length of the can. This dimension includes the chamfer length.

Uncut Length

Displays the uncut length of the can. This dimension includes the chamfer length.

Can Type

Displays the can type that was selected from the Catalog.

L2 Hull Length

Displays the calculated L2 hull length. This dimension is measured from the outside hull of the supporting member to the start of the chamfer on the tube section.

L2 Centerline Length

Displays the calculated L2 centerline length. This dimension is measured from the center of the supporting member to the start of the chamfer on the tube section.

L3 Hull Length

Displays the calculated L3 hull length.

L3 Centerline Length

Displays the calculated L3 centerline length.

Chamfer1 Length

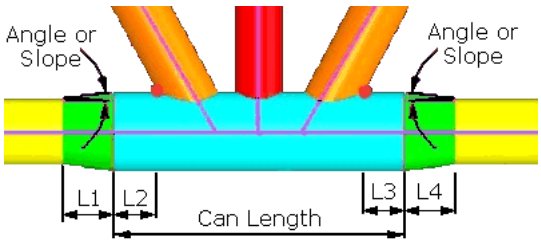
Displays the calculated Chamfer1 length.

Chamfer2 Length

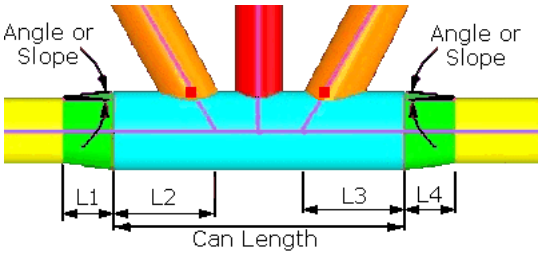
Displays the calculated Chamfer2 length.

Diameter - Tube

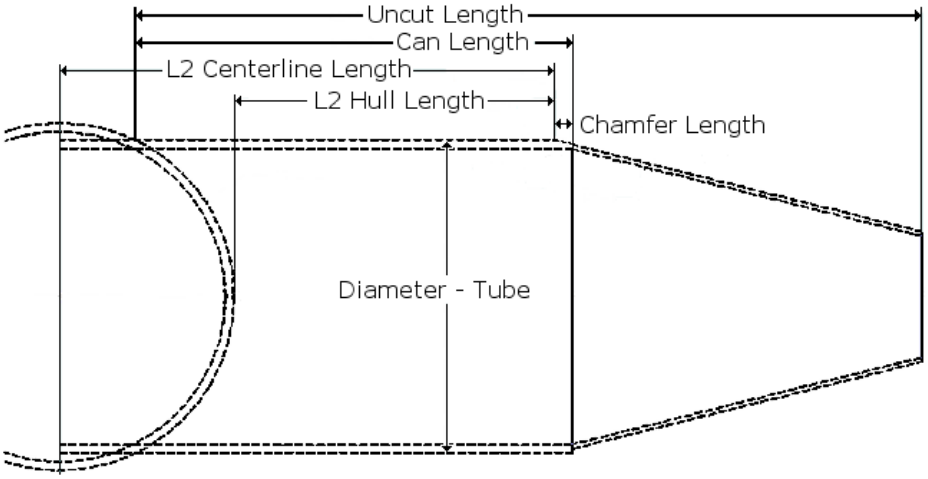
Displays the outside diameter of the tube part of the can.



L2 and L3 Lengths when using a Hull Extension Method



L2 and L3 Lengths when using a Centerline Method



Dimensions for Stub End Cans

Can Rule Tab (Can Properties Dialog Box)

Specifies the properties for the selected can rule.

Diameter Rule

Select how you want to define the diameter of the tube. You can select:

- **Outer Diameter** - Select this option to match the outside diameter of the tube with the outside diameter of the supporting leg.
- **Inner Diameter** - Select this option to match the inside diameter of the tube with the inside diameter of the supporting leg.

- **User Defined** - Select this option to specify the diameter of the tube yourself. Cones are created if you define a diameter that is significantly larger than the cross-section of the supporting leg.

Can Outside Diameter

Specifies the outside diameter of the can.

Can Inside Diameter

Specifies the inside diameter of the can.

Can Thickness

Specifies the plate thickness of the can plate.

Cone 1 Thickness

Specifies the plate thickness of cone 1.

Cone 1 Material

Select the material for cone 1.

Cone 1 Grade

Select the material grade for cone 1.

Can Material

Select the material for the can.

Can Grade

Select the material grade for the can.

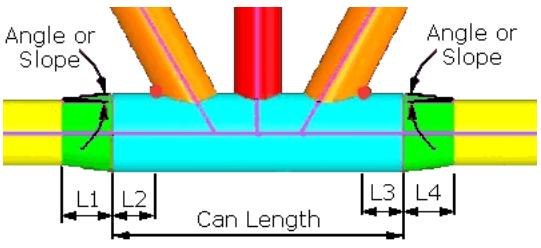
Minimum Extension Distance

Specify the minimum distance between the transverse member and the end of the can. If you type a distance that is less than the minimum extension distance that is defined in the Catalog, the software uses the distance defined in the Catalog, not the smaller distance that you specified here.

L2 Computation Method

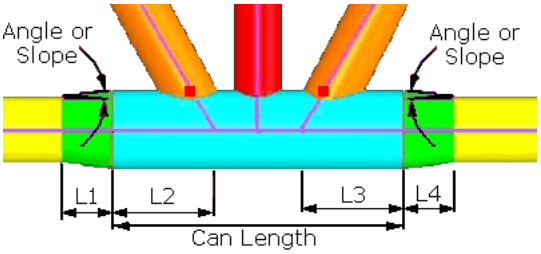
Specifies how you want Smart 3D to measure the L2 minimum distance.

- If you select **Hull-Factor**, the software calculates the outside diameter of the can times the value you type in the **L2 Factor** box. The result is used as the minimum L2 distance, which is measured from the corresponding end of the can to the point on the transverse member hull closest to that end of the can.
- If you select **Hull-Length**, the software uses the distance you specify in the **L2 Length** box for L2, which is measured from the corresponding end of the can to the point on the transverse member hull closest to that end of the can.



L2 and L3 Lengths when using a Hull Extension Method

- If you select **Centerline-Factor**, the software calculates the outside diameter of the can times the value you type in the **L2 Factor** box. The result is used as the minimum L2 distance, which is measured from the corresponding end of the can to the centerline of the transverse member closest to that end of the can.
- If you select **Centerline-Length**, the software uses the distance you specify in the **L2 Length** box for L2, which is measured from the corresponding end of the can to the centerline of the transverse member closest to that end of can.



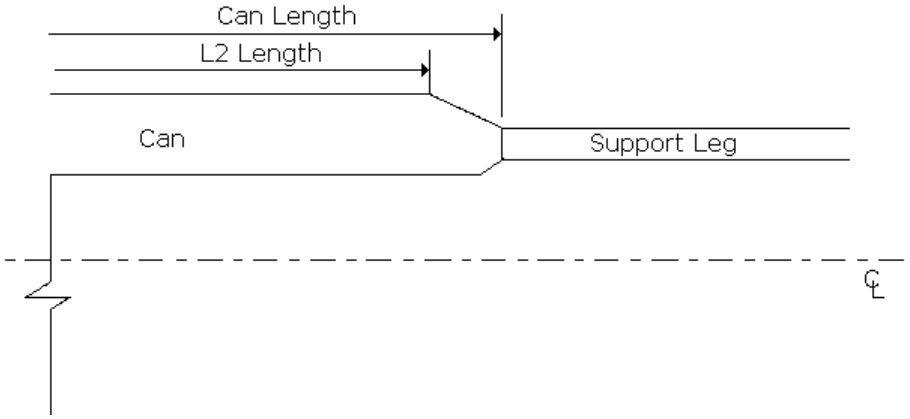
L2 and L3 Lengths when using a Centerline Method

L2 Factor

Type the factor to use to calculate L2. The default is **0.25**.

L2 Length

Type the L2 distance. This is the minimum distance that L2 can be. If there is no cone defined for that end of the can, then the L2 distance is the longer of the two chamfer lengths between the can and the support leg.

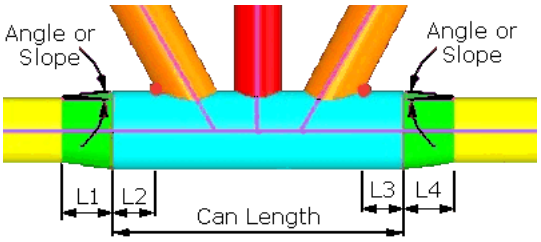


L3 Computation Method

Specifies how you want Smart 3D to measure the L3 minimum distance.

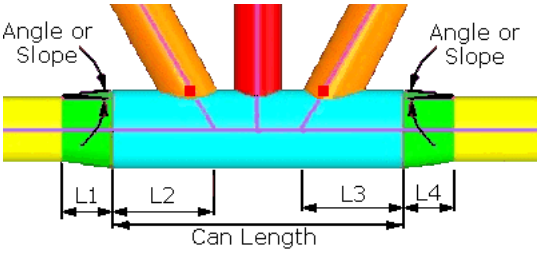
- If you select **Hull-Factor**, the software calculates the outside diameter of the can times the value you type in the **L3 Factor** box. The result is used as the minimum L2 distance which is measured from the corresponding end of the can to the point on the transverse member hull closest to that end of the can.

- If you select **Hull-Length**, the software uses the distance you specify in the **L3 Length** box for L3, which is measured from the corresponding end of the can to the point on the transverse member hull closest to that end of the can.



L2 and L3 Lengths when using a Hull Extension Method

- If you select **Centerline-Factor**, the software calculates the outside diameter of the can times the value you type in the **L3 Factor** box. The result is used as the minimum L3 distance, which is measured from the corresponding end of the can to the centerline of the transverse member closest to that end of the can.
- If you select **Centerline-Length**, the software uses the distance you specify in the **L3 Length** box for L3, which is measured from the corresponding end of the can to the centerline of the transverse member closest to that end of can.



L2 and L3 Lengths when using a Centerline Method

L3 Length

Type the L3 distance. This is the minimum distance that L3 can be. If there is no cone defined for that end of the can, then the L3 distance is the longer of the two chamfer lengths between the can and the support leg (see **L2 Length** above for an illustration).

L3 Factor

Type the factor to use to calculate L3. The default is **0.25**.

TIP Cone 1 is the cone nearest the start end of the supporting leg that the can is placed on.

Cone 1 Length Method

Specifies how you want to define the length of the first cone transition.

- If you select **Angle**, the length of the transition is determined by the geometry of the can diameter, the support member cross-section, and the angle specified in the **Cone 1 Angle** box. This is shown as Angle in the figures above.
- If you select **Distance**, the length of the transition is defined explicitly in the **Cone 1 Length** box. This distance is shown as L1 in the figures above.

- If you select **Slope**, the length of the transition is determined by the geometry of the can diameter, the support member cross-section size, and the slope specified in the **Cone 1 Slope** box. This is shown as Slope in the figures above.

Cone 1 Length

Specifies the length of the cone transition from the end of the support member to the start of the can tube. This distance is shown as L1 in the figures above. This option is available when **Cone 1 Length Method** is set to **Distance**.

Cone 1 Slope

Specifies the slope of the cone transition. The software does not create a cone if the slope you specify is greater than 100 (1/100) or less than 1^{-6}. This option is available when **Cone 1 Length Method** is set to **Slope**.

Cone 1 Angle

Specifies the angle of the cone transition. The software does not create a cone if the angle you specify is less than 1 degree or greater than 89.0 degrees. This option is available when **Cone 1 Length Method** is set to **Angle**.

TIP Cone 2 is the cone nearest the end of the supporting leg that the can is placed on.

Cone 2 Length Method

Specifies how you want to define the length of the end cone transition.

- If you select **Angle**, the length of the transition is determined by the geometry of the can diameter, the support member cross-section, and the angle specified in the **Cone 2 Angle** box. This is shown as Angle in the figures above.
- If you select **Distance**, the length of the transition is defined explicitly in the **Cone 2 Length** box. This distance is shown as L4 in the figures above.
- If you select **Slope**, the length of the transition is determined by the geometry of the can diameter, the support member cross-section size, and the slope specified in the **Cone 2 Slope** box. This is shown as Slope in the figures above.

Cone 2 Length

Specifies the length of the cone transition from the end of the support member to the start of the can tube. This distance is shown as L4 in the figures above. This option is available when **Cone 2 Length Method** is set to **Distance**.

Cone 2 Slope

Specifies the slope of the cone transition. The software does not create a cone if the slope you specify is greater than 100 (1/100) or less than 1^{-6}. This option is available when **Cone 2 Length Method** is set to **Slope**.

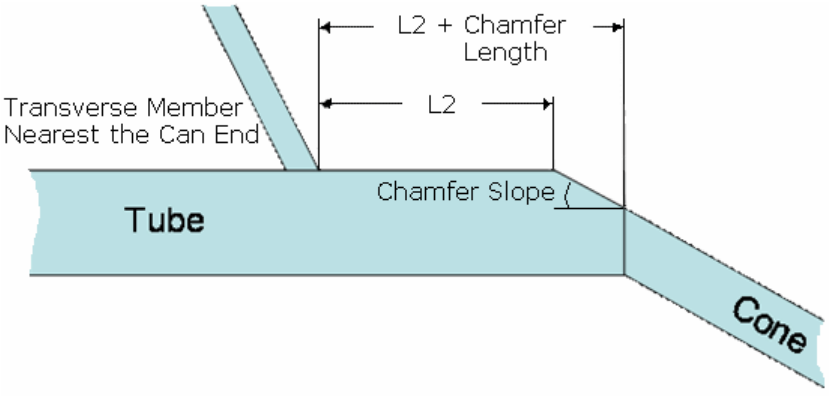
Cone 2 Angle

Specifies the angle of the cone transition. The software does not create a cone if the angle you specify is less than 1 degree or greater than 89.0 degrees. This option is available when **Cone 2 Length Method** is set to **Angle**.

Chamfer Slope

Specifies the slope of the chamfer located at the end of the can where the can's tube transitions into the can's cone. The chamfer length, which is determined by the defined sloped, contributes to the length of the can part of the overall can. The value you type here

is used for both ends of the can.



Roundoff Distance

Type the distance to round off the total can length. The length of the can is rounded up to the next increment of this value.

Cone 2 Thickness

Specifies the plate thickness of cone 2.

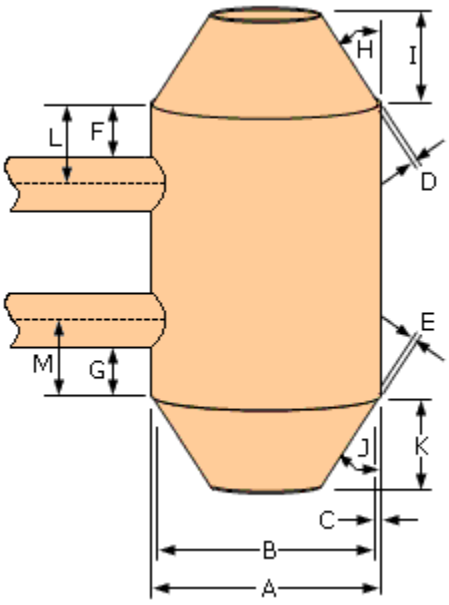
Cone 2 Material

Specifies the plate material of cone 2.

Cone 2 Grade

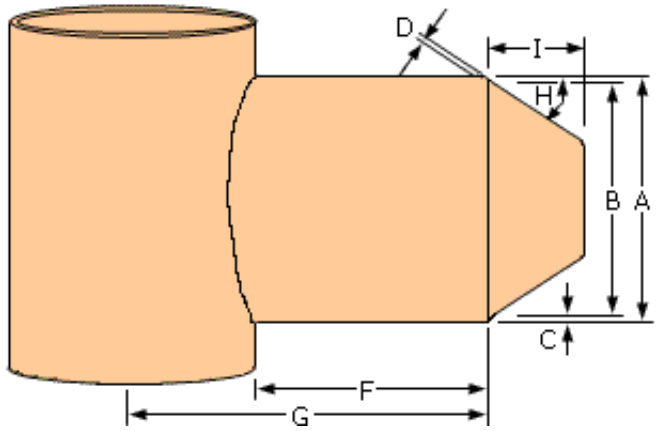
Specifies the plate material grade of cone 2.

Inline Can



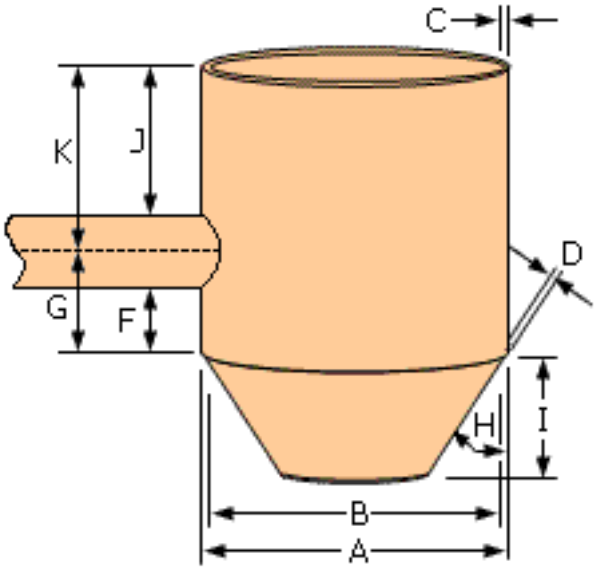
A=Can Outside Diameter
B=Can Inside Diameter
C=Can Thickness
D=Cone 1 Thickness
E=Cone 2 Thickness
F=L2 Length by Hull
G=L3 Length by Hull
H=Cone 1 Angle or Slope
I=Cone 1 Length
J=Cone 2 Angle or Slope
K=Cone 2 Length
L=L2 Length by Centerline
M=L3 Length by Centerline

Stub Can



A=Can Outside Diameter
B=Can Inside Diameter
C=Can Thickness
D=Cone Thickness
F=L2 Length by Hull
G=L2 Length by Centerline
H=Cone Angle or Slope
I=Cone Length

End Can



A=Can Outside Diameter
B=Can Inside Diameter
C=Can Thickness
D=Cone Thickness
F=L2 Length by Hull
G=L2 Length by Centerline
H=Cone Angle or Slope
I=Cone Length
J=L3 Length by Hull
K=L3 Length by Centerline

Cross Section Tab (Can Properties Dialog Box)

Displays the can properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one can and then selected the properties command, only the common properties between the selected cans display.

The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties.

Category

Select the properties that you want to view for the member. Currently, only Standard properties are available for BUCan designed member.

Standard

Name

Displays the name of the designed member.

Symbol Definition

Displays the symbol ProgID.

Definition

Specifies the definition rule for the part class. Do not change the default value.

Parameter Rule

Specifies the parameter rule for the part class. Do not change the default value.

Centroid X

Displays the location of the centroid along the local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Centroid Y

Displays the location of the centroid along the local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about X (Ixx)

Displays the moment of inertia for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about Y (Iyy)

Displays the moment of inertia for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Statical Moment (Sw)

Displays the warping statical moment. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about X (Sxx)

Displays the section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about Y (Syy)

Displays the section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Torsional Moment of Inertia (J)

Displays the torsional moment of inertia for the section. This property is read- only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Constant (Cw)

Displays the warping constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Flexural Constant (H)

Displays the flexural constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Polar Radius of Gyration about Shear Center (ro)

Displays the polar radius of gyration about the shear center. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about X axis (Rxx)

Displays the radius of gyration for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Y axis (Ryy)

Displays the radius of gyration for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Principle XY (Rxy)

Displays the radius of gyration about the principle xy-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about X (Zxx)

Displays the plastic section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about Y (Zyy)

Displays the plastic section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Is Hollow

Displays **True** if the cross-section is hollow, such as a tube or can. Displays **False** if the cross-section is solid, such as a bar, or open, such as an I-beam.

Is Symmetric About X

Displays **True** if the cross-section is symmetric about the local x-axis of the cross-section.

Is Symmetric About Y

Displays **True** if the cross-section is symmetric about the local y-axis of the cross-section.

Diameter - Start

Displays the diameter of the start cone.

Diameter - End

Displays the diameter of the end cone.

Length - Start Cone

Displays the length of the start cone, which is the cone closest to the start of the supporting member.

Length - End Cone

Displays the length of the end cone.

Section Properties

Displays how section properties are defined.

Section Name

Type the section name. The section name must be unique across the catalog.

Short Name

Type the short name for the section.

EDI Name

Type the Electronic Data Interchange name for the section. This name is used when translating sections through CIMsteel. This property is currently not used.

Area

Specifies the cross-section area for the object. This property is read-only. To change this value, edit the object properties in Catalog or in the corresponding workbook.

Depth

Displays the depth for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Width

Displays the flange width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Perimeter

Displays the outside perimeter distance for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Unit Weight

Displays the unit weight.

Length Extension

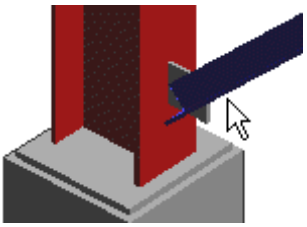
Displays the length extension, which is the minimum distance between the transverse member and the end of the tube part of the can.

Diameter - Tube

Displays the outside diameter of the tube part of the can.

SECTION 10

Place Assembly Connection



Places an assembly connection or free end cut at the selected frame connection. Assembly connections define the necessary trimming between member parts and provide for the generation of parts such as base plates, gusset plates, and clip angles. Assembly connections also control cutbacks, copes, notches, bolt holes, weld preparations, and slots.

This command places member assembly connections or a free end cut on the member. The software automatically places slab assembly connections when you place the slab.

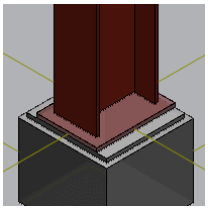
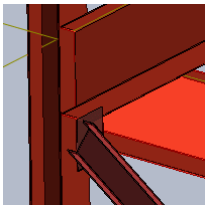
Member Assembly Connections

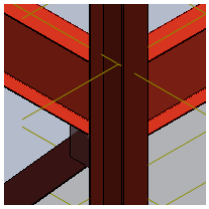
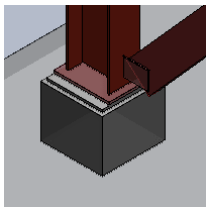
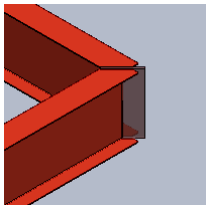
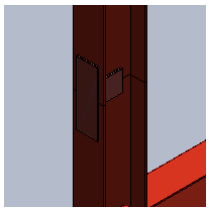
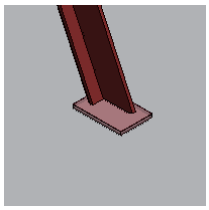
Member assembly connections are similar to frame connections, but define the necessary trimming between member parts and the generation of parts such as base plates, gusset plates, and clip angles. Assembly connections control member features including cutbacks, copes, notches, bolt holes, and slots. Whether or not features are placed depends on the member assembly connection type and the geometry of the connection between the members.

There are several basic assembly connections delivered with the software. You can create your own assembly connections by editing the **StructAssemblyConnections.xls** workbook, and then bulk loading the workbook. For more information on creating your own assembly connections, see the Structure Reference Data Guide.

NOTE For very complex nodal connections, instead of writing a custom assembly connection, you might want to try *Trim Members* (on page 201) in the Structure task.

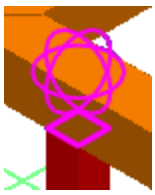
The following general assembly connections are delivered with the software:

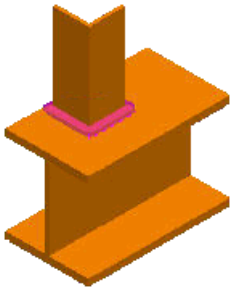
	The base plate assembly connection places a plate at the end of an unsupported member. This assembly connection requires an unsupported frame connection on one member. For example, use this assembly connection to place a base plate at the bottom of a column.
	The corner gusset plate assembly connection connects a vertical brace to a beam and column intersection using a gusset plate. This assembly connection requires a frame connection with three members, such as vertical corner brace.

	The fitted assembly connection connects two members. This assembly connection requires a frame connection with two members, such as axis, seated, or flush. Examples of this connection include a beam framing into a column or a beam framing into another beam.
	The gusset plate assembly connection connects a vertical or horizontal brace to a beam or a vertical brace to a column using a gusset plate. This assembly connection requires a frame connection with two members, such as axis.
	The miter assembly connection connects two members that meet at an angle but are co-planar. This assembly connection requires a frame connection with two members, such as axis. In addition, the members must be end connected.
	The splice assembly connection connects two members that are collinear and end connected. This assembly connection requires a frame connection with two members, such as axis.
	The general surface assembly connection connects a member end with a nonmember surface such as a slab, a deck, or a hull plate. The member is cut to surface and a base plate is placed on the member end.

End Cut Assembly Connections

The following end cut assembly connections are delivered with the software:

	The end cut along assembly connection cuts the end of one member to the sides or bottom of another member along the second member's length.

	The end cut bearing plate assembly connection connects the end of a member along the axis of another member. This is typically used to attach the end of a member to the flange of the supporting member. The bearing plate connection works at any angle and provides circular, rectangular, and triangular bearing plates depending on the supported member's shape.
	The end cut miter assembly connection connects two members that meet at an angle but are co-planar. This assembly connection required a frame connection with two members, such as axis. In addition, the members must be end connected.
	The end cut split assembly connection connects two members that are collinear and end connected. This assembly connection requires a frame connection with two members, such as axis.
	The end cut split long box assembly connection is used when the supported and supporting members are perpendicular to each other. The supporting member is cut square to the supported member while an item from the Axis category is placed on the supported member.
	The end cut split seam angle assembly connection is used when the supported and supporting members are: ▪ member axes are collinear to each other ▪ have identical cross-section type and size ▪ have identical cardinal point and rotation values
	The end cut split short box assembly connection is used when the supported and supporting members are perpendicular to each other. The supported member is cut square to the supporting member while an item from the Axis category is placed on the supporting member.

End Cut Assembly Naming Conventions

Each end cut assembly connection name consists of three parts:
[EndCutCase]_[WebCutType]_[FlangeCutType]

where:

- [EndCutCase]
  - **Axis** - Supported member is bounded by the axis curve of the supporting member
  - **LongBox** - End-to-end perpendicular supported member is extended, and an axis end cut is applied to the supporting member
  - **ShortBox** - End-to-end perpendicular supporting member is extended, and an axis end cut is applied to the supported member

- **Miter** - End-to-end non-collinear
- **Split** - End-to-end collinear
- **SeamAngle** - End-to-end collinear where one end is extended

- [WebCutType]

  The WebCutType is used to type of web cut that is to be applied at the top of the web and at the bottom of the web. The following combinations are available (W indicates a straight cut, C indicates a Cope cut, and S indicates a Snipe cut):

  - W#W#
  - W#C#
  - W#S#
  - C#C#
  - C#S#
  - C#W#
  - S#S#
  - S#C#
  - S#W#

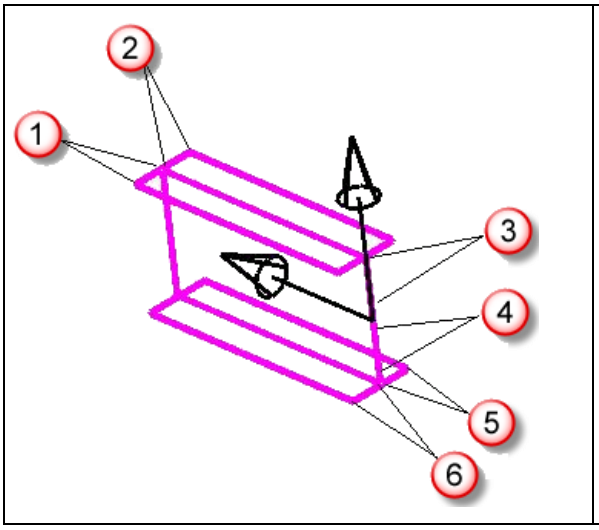
- [FlangeCutType]

  The FlangeCutType is used to type of flange cut that is to be applied at the left flange section and at the right flange section. The following combinations are available (W indicates a straight cut, C indicates a cope cut, and S indicates a snipe cut):

  - W#W#
  - W#S#
  - S#S#
  - S#W#

You cannot apply different flange cuts when the supported member has both a top flange and a bottom flange. In these cases, the same flange cut type is applied to both the top and bottom flange.

You cannot apply different web cuts or flange cuts in the end-to-end cases. The end-to-end cases place a web cut and flange cut on both the supported member and the supporting member. For these cases, the same web cut and flange cut are applied to both the supported member and the supporting member.

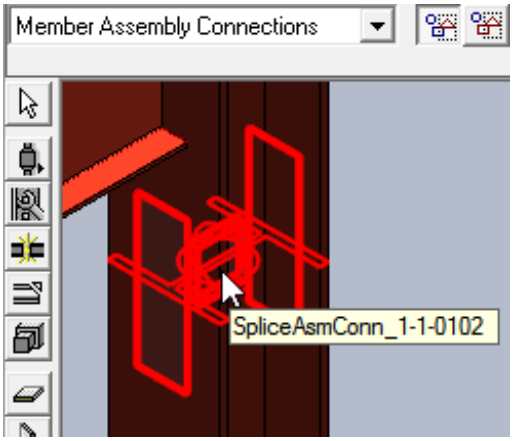


1 - Top left flange
2 - Top right flange
3 - Web top
4 - Web bottom
5 - Bottom right flange
6 - Bottom left flange

See *Members* (on page 22) in the Structure task for important related information.

Locating Assembly Connections

Assembly connections do not display in the model. However, if you set the **Locate Filter** to **Member Assembly Connections** in the Structure task, you can locate and select assembly connections for review and editing. Assembly connections are located at the ends of member parts and appear as circles when you move the cursor over them. Any assembly components, such as gusset plates, associated with the assembly connection also highlight.



When you select an assembly connection, the software displays the assembly connection type in the ribbon. Select the **Edit > Properties** command to edit the assembly connection properties. You cannot measure distances using the assembly connection graphic as an end point when using the **Tools > Measure** command.

Place Assembly Connection Ribbon

Specifies the properties for the assembly connection that you are placing or editing.

Assembly Connection Properties

Activates the **Assembly Connection Properties** dialog box. You can use this dialog box to

specify additional properties that you cannot set on the ribbon. For more information, see *Assembly Connection Properties Dialog Box* (on page 184).

Select Member/Connection

Activated automatically by the software so that you can select the frame connection or the member for which you want to place assembly connections.

Finish

Click to place the assembly connection the model.

Cancel

Rejects the selected object.

Accept

Confirms that the selected members, is the member to place assembly connections for. The software displays in tentative mode the results of the assembly connection.

Condition

Specifies how you want the software to handle existing assembly connections when you try to place a new assembly connection at the same location. Select **Retain existing** to keep the existing assembly connection. Select **Update existing** to replace the existing assembly connection with the new assembly connection.

By Rule

Select to allow the software to select the assembly connection to use based on the selected member parts and their orientation to each other.

Type

Select the assembly connection type to use. If you select **By Rule**, the software determines the correct assembly connection to use based on the geometry between the member parts, and this option is unavailable. If you select **More**, all available assembly connections display from which you can select the assembly connection to use. For more information about assembly connections, see *Member Assembly Connections* (on page 29).

Member Part Parent

Select to make the member part the parent object of the assembly connection. Clear this option to specify a generic system, using the **System** box, to be the assembly connection parent. Assembly connections created using member parts as their parents can be edited to have a system parent later if needed.

System

Select the system to which the assembly connection belongs. You can define new systems in the Systems and Specifications task. This option is unavailable if you select **Member Part Parent**. Assembly connections created using a system as their parent can be edited to have a member part as their parent later if needed.

Name

Specify the name of the assembly connection.

Modify Auxiliary Parts Ribbon

Displays the options that you use to add auxiliary parts to the selected assembly connection.

Properties

Activates the **Assembly Connection Properties Dialog Box**, which you use to view and modify the properties of the assembly connection before the assembly connection is committed to the database. For more information, see *Assembly Connection Properties Dialog Box* (on page 184).

Assembly Connection Smart Item

Displays valid smart items for the selected assembly connection.

Add/Delete Auxiliary Parts

Select the parts that can be connected to the selected assembly connection and do not already have an existing assembly connection. Parts that do not have common geometry with the selected assembly connection cannot be selected.

Accept

Adds parts from selected parts to the assembly connection and saves the changes to the database.

Reject

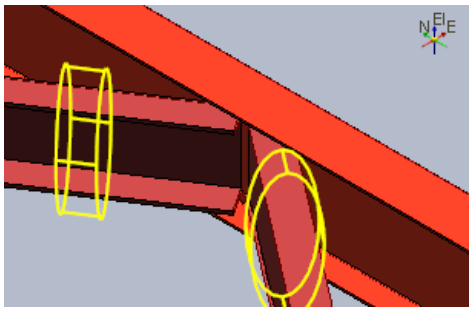
Removes any selected parts from the auxiliary parts list. Only selections made after the last commit are removed.

What do you want to do?

- *Place an assembly connection* (on page 181)
- *Edit assembly connection properties* (on page 182)
- *Delete an assembly connection* (on page 182)
- *Add auxiliary parts to an assembly connection* (on page 182)
- *Edit assembly connection parts* (on page 183)
- *Place free end cuts* (on page 183)
- *Edit free end cut properties* (on page 183)
- *Delete free end cut* (on page 184)

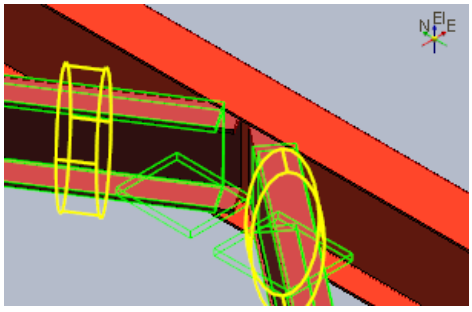
Place an assembly connection

1. Click **Place Assembly Connection** on the vertical toolbar.
2. Select the **By Rule** option if you want the software to automatically select the type of assembly connection to place. Clear the **By Rule** option to select the assembly connection type yourself.
3. Select the frame connection nearest the member end to which to apply the assembly connection.

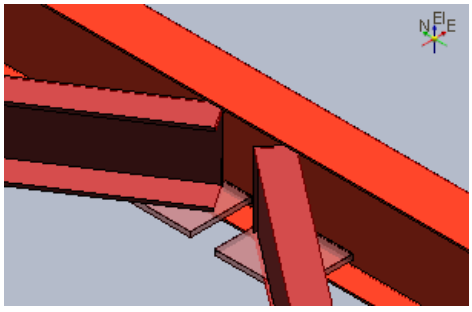


4. If you are selecting the assembly connection type yourself, use the **Type** option.
5. Click **Accept** ✓.

   The software automatically selects the assembly connection and displays the results.



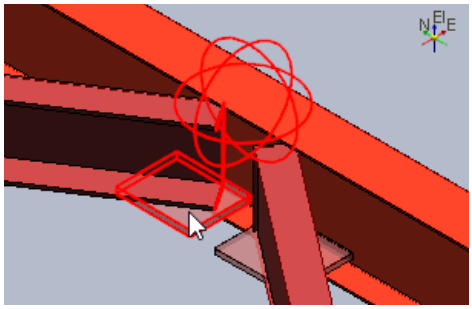
6. Click **Finish**.



NOTE For more information about the types of assembly connections, see *Member Assembly Connections* (on page 29).

Edit assembly connection properties

1. Click **Select** on the vertical toolbar.
2. Select **Member Assembly Connections** in the **Locate Filter**.
3. Select the assembly connection to edit.



4. Click **Edit > Properties**.
5. Edit the assembly connection properties as needed. For more information, see *Assembly Connection Properties Dialog Box* (on page 184).

Delete an assembly connection

1. Click **Select** on the vertical toolbar.
2. Select **Member Assembly Connections** in the **Locate Filter**.
3. Select the assembly connection to delete.
4. Click **Delete**.

Add auxiliary parts to an assembly connection

This command supports adding parts to an existing assembly connection by selecting an adjacent plate, profile or member parts that have overlapping geometry.

1. Select an existing assembly connection.
2. Click **Select Auxiliary Parts** on the ribbon bar.

   NOTE The **Select Auxiliary Parts** is enabled for any assembly connection. You can only add valid auxiliary parts to an assembly connection.

3. Click **Accept** to add all intersecting parts on the selected parts to the assembly connection.

Edit assembly connection parts

1. Select an assembly connection, and click **Select Auxiliary Parts** on the ribbon bar.
2. Select the parts not already connected to the selected assembly connection that have intersecting geometry.
3. To remove a single part, re-select it from the graphic view or **Workspace Explorer**.
4. To remove the auxiliary parts added in the current command execution, click **Reject** on the ribbon bar.
5. Click **Accept** to add all intersecting parts on the selected parts to the assembly connection.

Place free end cuts

1. Click **Place Assembly Connection** on the vertical toolbar.
2. Select the unsupported frame connection nearest the member end to which you want to apply the free end cuts.

   If you select a member part, the software automatically places free end cuts at the unsupported frame connections.
3. Click **Accept.**

   The software automatically selects the free end cuts and displays the results.
4. Click **Finish**.

Edit free end cut properties

1. Click **Select** on the vertical toolbar.
2. Select **Member Assembly Connections** in the **Locate Filter** box.
3. Select the free end cut to edit.
4. Click **Edit > Properties**.
5. Edit the free end cut properties as needed. For more information, see Free End Cut Properties Dialog Box.

Delete free end cut

1. Click **Select** on the vertical toolbar.
2. Select **Member Assembly Connections** in the **Locate Filter**.
3. Select the free end cut to delete.
4. Click **Delete**.

Assembly Connection Properties Dialog Box

Specifies the properties for the member assembly connection that you are editing.

Occurrence Tab (Assembly Connection Properties Dialog Box) (on page 184)
Definition Tab (Assembly Connection Properties Dialog Box) (on page 189)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)
Selection Tab (Assembly Connection Properties Dialog Box) (on page 189)

See Also

Edit assembly connection properties (on page 182)

Occurrence Tab (Assembly Connection Properties Dialog Box)

The **Occurrence** tab displays the member assembly connection properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid, and the corresponding property value appears on the right side of the grid. If you selected more than one assembly connection, and then selected the properties command, only the common properties between the selected assembly connections display.

When viewing properties for a single assembly connection, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the Structure Reference Data Guide for more information on properties.

Name

Displays the name of the assembly connection. The assembly connection name is based on the **Name Rule** selection. To type a new name for the assembly connection, in the **Name Rule** box, select **User Defined**, and then type a name for the assembly connection in the **Name** box.

Name Rule

Specify the naming rule to use to name this assembly connection.

- **Default Name Rule** - Names the assembly connection using this format: <assembly connection>-<location>-<unique index> where <assembly connection> is the name of the assembly connection defined in the catalog, <location> is the global workshare location, and <unique index> is an index number that starts at 0001. For example, GussetPlateAsmConn_1-1-0045.
- **Unique Name Rule** - Names the assembly connection using this format: <parent system name>-<assembly connection>-<location>-<unique index> where <parent

system name> is the parent System selected for the assembly connection, <assembly connection> is the name of the assembly connection defined in the catalog, <location> is the global workshare location, and <unique index> is an index number that starts at 0001. For example, Structure System-GussetPlateAsmConn_1-1-0045.

- **User Defined** - Select to specify the assembly connection name yourself in the **Name** box.

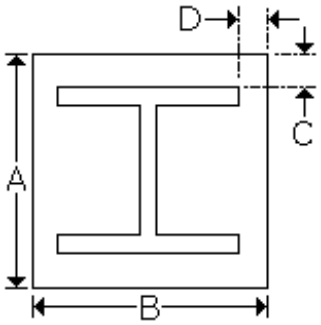
System

Select the system to which the assembly connection that you are placing belongs. You can create new systems in the Systems and Specifications task.

Base Plate Assembly Connection Properties

Depth Clearance

Specify the clearance between the flange of the member and the edge of the base plate. This is dimension C in the figure.



Width Clearance

Specify the clearance between the flange of the member and the edge of the base plate. This is dimension D in the figure.

Sizing Rule

Select the sizing rule method for the base plate.

Plate Category

Select the plate category.

Plate Type

Select the plate type.

Miter Assembly Connection Properties

Top Distance

Specifies the distance between the top flange of the member section and the top of the plate.

Bottom Distance

Specifies the distance between the bottom flange of the member section and the bottom of the plate.

Left Distance

Specifies the distance between the left edge of the member section and the left edge of the plate.

Right Distance

Specifies the distance between the right edge of the member section and the right edge of the plate.

Symmetry

Controls how to cut back the member ends when the clearance value is not zero. Select **Center** to specify that both members are cut back equally. Select **Right** to specify that the

first member that you selected be cut back. Select **Left** to specify that the second member that you selected be cut back.

With Plates

Specifies whether or not a plate is inserted between the member ends. Select **False** to not place the plate. Select **True** to place the plate.

Clearance

Specifies the distance between the member ends.

Sizing Rule

Select the sizing rule method for the base plate.

Slab by Member Boundary

Clearance

Type a clearance distance between the edge of the slab and the member.

Port Face Position

Select the location on the member at which the slab is to stop. You can select the outmost plane, the centerline, or the in-most plane on the member.

Detailed Connection

Select **True** to trim the slab using the boundary member as the cutting edge. Select **False** to not trim the slab.

Offset

Specify the distance between the selected **Port Face Position** and the edge of the slab. A negative value moves the edge into the body of the slab. A positive value moves the edge out from the body of the slab.

Slab by Slab Assembly

Trim Type

Defines the type of trimming.

- **By Whole Slab** - Trim by the slab boundary geometry before trim.
- **By Lateral Face** - Trim by the slab boundary geometry after trim.
- **By Extended Lateral Face** - Trim by the extension of the lateral face of the slab boundary.

Trim Excess Material

Specify whether to trim the exceeding part.

Trim Excess Material Offset

Specify the offset applied on the lateral face used to trim.

Slab Free Edge Assembly

Reference Direction

Select the reference direction for the angle:

Normal

The angle is measured from a vector perpendicular to the slab edge.

- **Horizontal** - The angle is measured from the global XY plane in the model.
- **Vertical** - The angle is measured from the Z-Axis in the model.

Angle

Type a slope for the slab edge represented by the assembly connection. If the slab was place using the **Face Position Top**, the side face rotates about the top slab edge. If the slab was placed using the **Face Position Bottom**, the side face rotates about the bottom slab edge.

Offset

Specify the distance between the selected boundary object and the edge of the slab.

Splice Assembly Connection Properties

Symmetry

Controls how to cut back the member ends when the clearance value is not zero. Select **Center** to specify that both members are cut back equally. Select **Right** to specify that the first member that you selected be cut back. Select **Left** to specify that the second member that you selected be cut back.

Clearance

Specifies the distance between the ends of the members.

Splice With

Select the plates that you want to use in the splice.

Web Plate Position

Select a web plate position.

Distance from flange gage line

Specifies the distance from the flange gage line.

Distance from web gage line

Specifies the distance from the web gage line.

Flange Plate Thickness

Specifies the thickness of the flange plates.

Flange Plate Length

Specifies the length of the flange plates.

Flange Plate Width

Specifies the width of the flange plates.

Flange Plate Category

Select the plate category for the flange plates.

Flange Plate Type

Select the plate type for the flange plates.

Web Plate Thickness

Specifies the thickness of the web plates

Web Plate Length

Specifies the length of the web plates.

Web Plate Width

Specifies the width of the web plates.

Web Plate Category

Select the plate category for the web plates.

Web Plate Type

Select the plate type for the web plates.

General Surface Assembly Connection Properties

With Pad

Select **True** to place a steel pad between the member end and the surface. Select **False** to have the member end connect directly to the surface.

Pad Type

Select the shape of the pad.

Sizing Rule

Select whether you want to the software to automatically size the pad or if you want to size the pad.

Offset

Type the distance between the end of the member and the face of the surface.

See Also

Assembly Connection Properties Dialog Box (on page 184)

Definition Tab (Assembly Connection Properties Dialog Box)

The **Definition** tab displays the member assembly connection properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one assembly connection and then selected the properties command, only the common properties between the selected assembly connections display.

The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties.

See Also

Assembly Connection Properties Dialog Box (on page 184)

Selection Tab (Assembly Connection Properties Dialog Box)

Specifies the rule criteria used for the member assembly connection. Assembly connection rule selections affect the selections available for other smart occurrence objects that are children of the assembly connection.

TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.
- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.
- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.



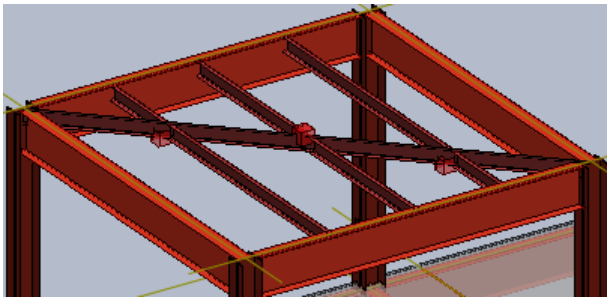
1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.
2. Click the **Rule Based** box again to select and set all objects to rule based.

IMPORTANT

- The available questions, answers, and results vary depending upon the detailed parts associated with the connection.
- The questions in each **Selection** tab represent the default rules delivered with the software. Customized rules may have different questions.

SECTION 11

Place Splits



Divides a member system (created in the Structure task) into multiple member parts. The resulting member system is a set of continuous member parts that move as a single entity. This splitting is useful when you want to resolve the interference between two intersecting structural objects, such as pair of cross braces, or when you want to split the columns in your model at certain elevations. You can split member systems at another member system, at a point along the member system, a grid plane, an elevation plane, deck, bulkhead, or plate system.

Only member systems can be split using the command. However, the object used to split the member system can be another member system, a grid plane, an elevation plane, deck, bulkhead, or plate system.

To reconnect a split member system, delete the split connections. The split connects are child objects of the member system that they are splitting.

If after you split a member system into multiple member parts you want to delete an individual member part, you will need to create a member system parent for that member part using the **Convert** option. Set the locate filter to **Parts** and select the member part, then click **Convert** on the modify ribbon. For more information, see *Modify Linear Member Part Ribbon* in *Place Linear Member Systems* (on page 35).

Member End Releases

If a column is split, the upper end of the lower column and the lower end of upper column has all end releases fixed. The lower end of the lower column and the upper end of the upper column do not change.

For beams and braces that are split, end releases on either side of the split are pinned for rotation (RY and RZ). The free end of a cantilevered beam or brace has fully fixed member end releases.

Boundary Conditions

Boundary Conditions remain on the ends for which they were defined. For example, if end 1 of the original member was fully supported (X, Y, Z, RX, RY, RZ) and end 2 was supported only in RX and RZ, then end 1 of new member 1 gets fully supported and end 2 of new member 2 gets the RX and RZ support. The new interior ends (end 2 for member 1 and end 1 for member 2) do not have any supports defined.

For more information about boundary conditions, refer to the Structural Analysis task documentation.

Loads

Concentrated loads stay where they were placed (the physical location along the original member part length). The absolute or relative placement value is recalculated based on the new member part length.

Distributed loads are split into two distributed loads with the same magnitude (one distributed load for each new member part).

Partially distributed loads do one of two things based on the split location. If the split location is between the partially distributed load's end points, the load is split into two partially distributed loads - one on each new member part on either side of the split location.

If the split location is outside of the partially distributed load's end points, the load is unaffected other than having the end points' absolute or relative placement values recalculated based on the new member part length.

For more information about loads, refer to the Structural Analysis task documentation.

Fireproofing

When a member with fireproofing is split, Smart 3D creates new fireproofing on the two new split members with the same setback references as the original member's fireproofing. In addition, the software adjusts the setback distances so that the fireproofing maintains the same geometric start and end position and looks solid throughout.

If you delete a split, the fireproofing is merged using the properties from the existing fireproofing. You must edit the resulting fireproofing and adjust the setbacks and other properties as needed.

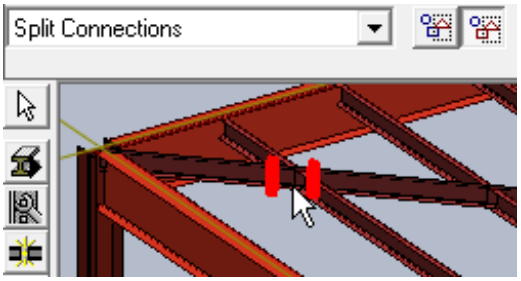
Designed Members

When splitting designed members, you need to be aware of these limitations:

- You can split designed members only by a rolled member, a specified point, a grid plane, or another designed member.
- If you split the designed member by a rolled member, then each plate in the designed member has a logical connection to the rolled member.
- If you split the designed member by a grid plane, then the plate parts on one side of the grid plane have a logical connection to the designed member part on the other side of the grid plane.
- The split always places a square end-cut on the designed member. This square cut is replaced by the appropriate end-cut when the designed member is detailed.
- Hangers, sketched boundaries, openings, features, chamfers, detailed plate parts, intersection seams connected to the designed member's plate systems *before* the split are deleted when the split is complete. Child plate systems and other plate systems referencing a designed member's plate system are put on the To Do List. You must redefine the reference relationships to remove them from the To Do List.
- Plate systems bounded by the designed member's plate systems before the split become unbounded when the split is complete. Stiffeners and edge reinforcements on these plate systems are also deleted upon completion of the split.

Locating Split Connections

Split connections do not display in the model. However, if you set the **Locate Filter** to **Connections**, you can locate and select split connections for review and editing. Split connections are located at the ends of member parts where the split is located and appear as circles when you move the cursor over them.



When you select a split connection, the software displays the parent member system of the split connection in the ribbon. Select the **Edit > Properties** command to edit the split connection properties. You cannot measure distances using the split connection as an end point when using the **Tools > Measure** command.

Place Split Ribbon

Specifies the properties for the split that you are placing or editing.

Split Properties

Activates the **Split Properties** dialog box. For more information, see *Split Connection Properties Dialog Box* (on page 199).

Place

Select how to define the split location.

- Select **By Object** to split the member system by another model object that you identify.
- Select **By Point** to split the member system at a point along the member system that you identify.

Finish

Splits the member systems using the options that you have specified.

Cancel

Clears the selected objects.

Accept

Accepts the selected objects and moves you to the next step in the command.

By Object Options

Split Members

Select the members to split. Remember, if you set **Split Status** to **By Rule** then you cannot split members that have **Continuous** set for the **Continuity Type**. You can split only members set to **Intercostal**.

Splitting Members

Select the objects at which to split the member systems. The objects can be another member system, a grid plane, an elevation plane, deck, or a bulkhead.

Split Status

Specifies the method to determine which member system splits another member system.

- **By Rule** - The software looks at the **Continuity Type** and **Continuity Priority Number** values defined on the *Member System Tab (Member System Prismatic Properties Dialog Box)* (on page 61) to determine which member system to split. Member systems set to **Continuous** split the other member system. Member systems set to **Intercostal** are split by the other member system. The **Continuity Priority Number** is used to select which member system is split when two member systems intersect, but both have **Intercostal** for the **Continuity Type**. Member systems with a lower continuity priority (1, 2, 3, for example) split member systems with a higher continuity priority (7, 8, 9, for example).
- **Split First** - The member systems that you identified during the **Split Members** step are split by the member systems that you identified during the **Splitting Members** step.
- **Split Second** - The member systems that you identified during the **Splitting Members** step are split by the member systems that you identified during the **Split Members** step.
- **Split Both** - All member systems that you have identified in both steps are split against each other.
- **Split None** - No member systems are split, but the interference between the two objects is suppressed.

By Point Options

Split Member

Select the member to split.

Splitting Point

Defines the split location point.

Location Definition

Specifies how the split location is calculated. The location can be defined as **Ratio** or **Distance**.

- A ratio location can be thought of as a distance percentage along the member. For example, a split placed at a relative distance of 0.333 is located at one-third the member length measured from the member start. A split placed at a relative distance of 0.5 is located at the middle of the member.
- A distance value is the actual distance from the start or end of the member. The software verifies that the absolute distance that you specify is not past the end of the member.

Offset Reference

Select whether to measure the split location from the start or the end of the member.

Offset Value

Specify the position of the split as either the ratio distance along the member or the absolute distance (as defined by **Location Definition**.)

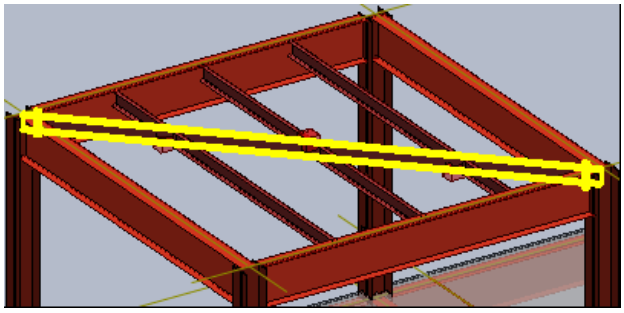
What do you want to do?

- *Split a member that intersects another member* (on page 195)
- *Split a cross brace* (on page 196)
- *Split columns at a plane* (on page 197)
- *Split a member at a point along member* (on page 198)
- *Set member split priority* (on page 199)
- *Remove a split* (on page 199)

Split a member that intersects another member

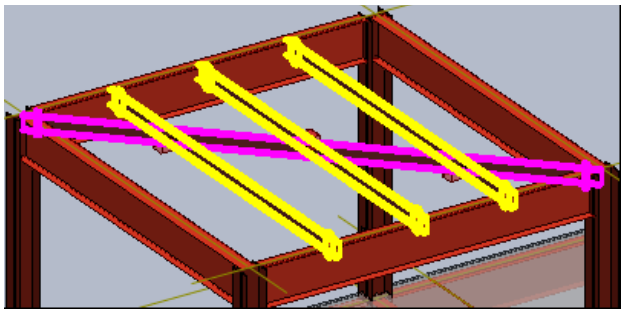
1. Click **Place Split** on the vertical toolbar.
2. In the **Place** box, select **By Object**.
3. In the **Split Status** box, select **Split First**.

   TIP If the **Continuity Type** and **Continuity Priority Number** values have been set for the member systems, you can use the **By Rule** option to have the software automatically determine which member system to split. For more information, see *Place Split Ribbon* in *Place Splits* (on page 191).
4. Select the member to split.

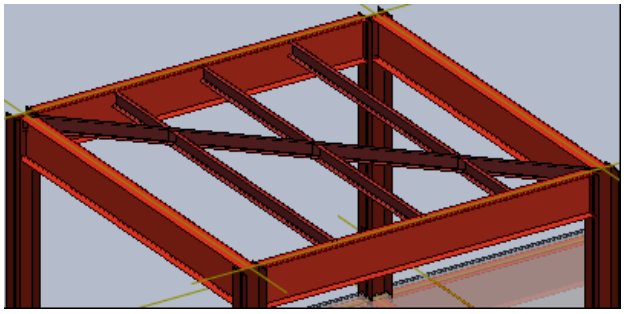


5. Click **Accept** ✓.

6. Select the splitting member.

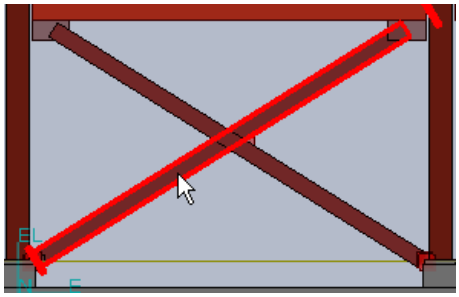


7. Click **Finish**.



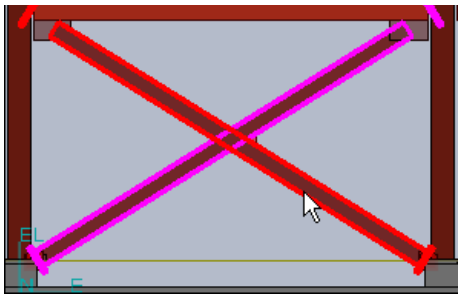
Split a cross brace

1. Click **Place Split** on the vertical toolbar.
2. In the **Place** box, select **By Object**.
3. In the **Split Status** box, select **Split First**.
4. Select the cross brace to split.

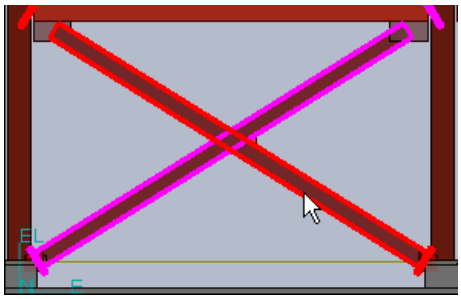


5. Click **Accept** ✓.

6. Select the other cross brace.

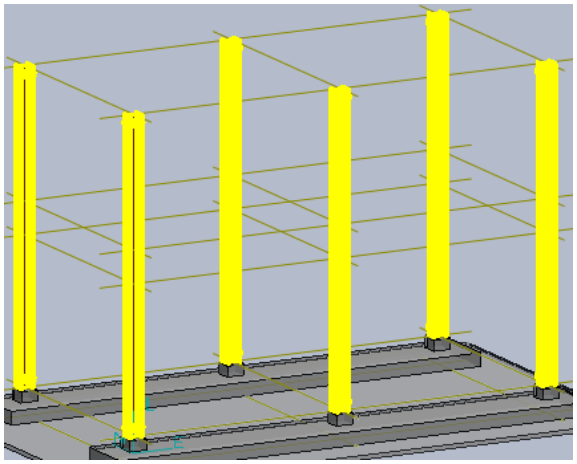


7. Click **Finish**.



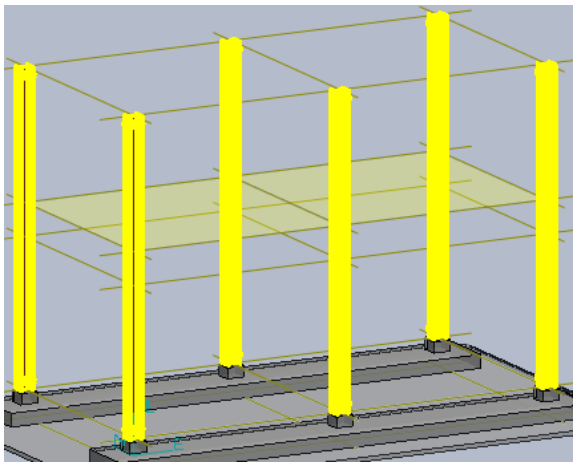
Split columns at a plane

1. Click **Place Split** on the vertical toolbar.
2. In the **Split Status** box, select **By Rule**.
3. Select the columns to split.



4. In the **Workspace Explorer**, select the elevation plane at which to split the columns.

TIP You can also select the elevation plane by selecting it on the Z-axis ruler. Select **View > Rulers** to turn on rulers.



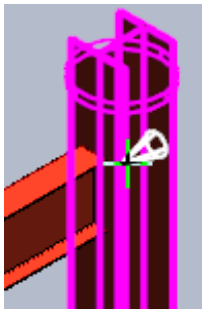
5. Click **Accept** ✓.
6. Click **Finish**.

NOTE Use the Grids task to create elevation planes.

Split a member at a point along member

1. Click **Place Split** on the vertical toolbar.
2. In the **Place** box, select **By Point**.
3. Select the member to split.

   An arrow indicates the split point.



4. In the **Location Definition** box, select **Distance** or **Ratio**.
5. Define the location for the split using the **Offset Reference** and **Offset Value** boxes.
6. Click **Finish**.

Set member split priority

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Members System**.
3. Select the member system to edit.
4. Click **Properties**.
5. Set the **Continuity Type** and **Continuity Priority Number** options as required. See *Member System Tab (Member System Prismatic Properties Dialog Box)* (on page 61) for information on these two properties.
6. Click **OK**.

Remove a split

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Split Connections**.
3. In a graphic view, select the split to remove.
4. Click **Delete**.

Split Connection Properties Dialog Box

Specifies the properties for the split connection that you are editing.

See Also

General Tab (Split Connection Properties Dialog Box) (on page 199)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

General Tab (Split Connection Properties Dialog Box)

Specifies the properties for the split connection.

Category

Select the type of properties to view for the selected split connection.

Name

Specifies the name of the split connection.

Name Rule

Select the name rule to use to name the split connection.

- **Default Name Rule** - Names the split connection using the format: <split status>-<location>-<index> where <split status> is the Split Status that was used to create the split connection, <location> is the global workshare location, and <index> is a unique index number that starts at 0001. For example, Split First-1-0004.

- **Unique Name Rule** - Names the split connection using the format: <split status>-<location>-<index> where <split status> is the Split Status that was used to create the split connection, <location> is the global workshare location, and <index> is a unique index number that starts at 0001. For example, Split First-1-0004.
- **User Defined** - Select this name rule to name the split connection yourself in the **Name** box.

Parent

Displays the parent member system to which the split belongs.

Split Parent Status

Specifies the method Smart 3D uses to determine the member system that splits another member system. The list is defined by the SplitStatus codelist.

By Rule indicates that the software looks at the **Continuity Type** and **Continuity Priority Number** values defined on the *Member System Tab (Member System Prismatic Properties Dialog Box)* (on page 61) to determine the member system to split. Member systems set to **Continuous** split the other member system. Member systems set to **Intercostal** are split by the other member system. The **Continuity Priority Number** specifies the member system that is split when two member systems intersect, but both have **Intercostal** for the **Continuity Type**. Member systems with a lower continuity priority (1, 2, 3, for example) split member systems with a higher continuity priority (7, 8, 9, for example).

Split First indicates that the member systems that you identified during the **Split Members** step are split by the member systems that you identified during the **Splitting Members** step.

Split Second indicates that the member systems that you identified during the **Splitting Members** step are split by the member systems that you identified during the **Split Members** step.

Split Both indicates that all member systems that you have identified in both steps are split against each other.

Split None indicates that no member systems are split, but the interference between the two objects is suppressed.

Offset

Specify the offset from the intersecting plane or surface to place the split. You can specify a positive or negative number. This option is only available after the split connection is placed when defining a split using a grid plane, elevation plane, or surface as the splitting object.

See Also

Split Connection Properties Dialog Box (on page 199)

SECTION 12

Trim Members

Manually copes and snips member parts based on objects and planes that you specify. Use this command when assembly connections are not sufficient to resolve the interference conflicts due to the position, orientation, and complexity of the intersecting members. You can place a trim and an assembly connection at the same member end. However, the results of the trim might be overwritten by the assembly connection results making the trim redundant. Both the trim and the assembly connection are listed in the **Workspace Explorer** under the member part.

Trim Member Ribbon

Specifies the properties for the trim that you are placing or editing.

Trim Feature Properties

Activates the **Trim Feature Properties** dialog box. You can use this dialog box to specify additional properties that you cannot set on the ribbon. For more information, see *Trim Feature Properties Dialog Box* (on page 204).

Select Members

Select the members that you want to cut.

Select Cutters

Select the objects to use to cut the members.

Finish

Trims the members using the cutters you have defined.

Cancel

Rejects the selected objects.

Accept

Confirms that the selected objects are the one you want use.

Trim Feature

Select the type of trim you want to place. Click **More** to select from all available trimming types defined in the catalog.

Name

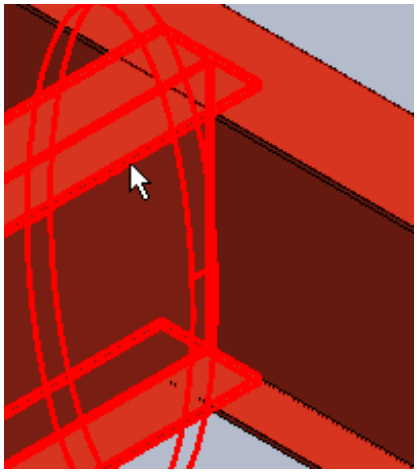
Specify the name of the trim feature.

What do you want to do?

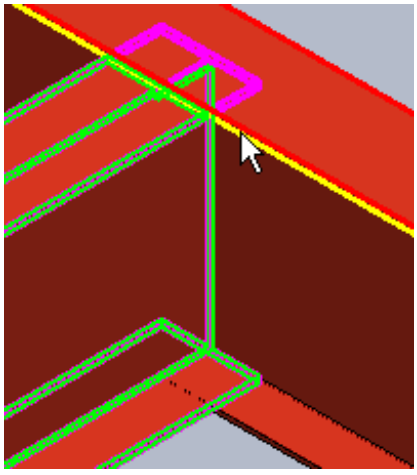
- *Cope a member web* (on page 202)
- *Trim member to surface* (on page 203)
- *Delete a member trim* (on page 203)

Cope a member web

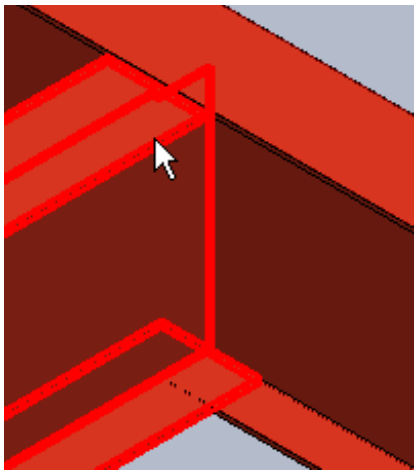
1. Click **Trim Member** on the vertical toolbar.
2. In the **Trim Feature** box on the ribbon, click **More**.
3. In the tree view, expand **Member End Trim Feature > Web Corner Cope Feature**.
4. Select **Web Cope 1/2in Clearance** in the **Name** column.
5. Click **OK**.
6. Select the member whose web you want to cope.



7. Click **Accept** ✓.
8. Select the plane that defines the cut into length of the web.



9. Click **Finish**.



Trim member to surface

1. Click **Trim Member** on the vertical toolbar.
2. In the **Trim Feature** box on the ribbon, click **More**.
3. In the tree view, expand **Member End Trim Feature > Surface Trim Feature**.
4. Select **SurfaceTrim_1** in the **Name** column.
5. Click **OK**.
6. Select the member to trim.
7. Click **Accept**.
8. Select the surface to trim the member.
9. Click **Finish**.

Delete a member trim

1. Click **Select** on the vertical toolbar.
2. Select **Structure** in the **Locate Filter**.
3. In the **Workspace Explorer**, expand the member prismatic part branch.
4. Select the trim feature node under the member part branch to delete.
5. Click **Delete**.

NOTE You can also use the **Tools > Select by Filter** command to select the member trims to delete. Define your filter using the **Object Type** tab and the **Structure > Features > Surface Trim Feature** or **Structure > Features > Web Corner Cope Feature** objects.

Trim Feature Properties Dialog Box

Specifies the properties for the trim feature that you are placing.

See Also

Definition Tab (Trim Feature Properties Dialog Box) (on page 204)

Definition Tab (Trim Feature Properties Dialog Box)

The **Definition** tab displays the trim feature properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one trim feature and then selected the properties command, only the common properties between the selected trim features display.

The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties.

See Also

Trim Feature Properties Dialog Box (on page 204)

Occurrence Tab (Feature Properties Dialog Box)

The **Occurrence** tab displays the feature properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one feature, and then selected the **Properties** command, only the common properties between the selected feature display.

When you view properties for a single feature, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the Structure Reference Data Guide for more information on properties.

Standard

Name

Displays the name of the feature. The feature name is based on the **Name Rule** selection. If you type a name in this field, the **Name Rule** property updates to **User Defined**.

Name Rule

Specifies the naming rule that you want to use to name this feature. You can select one of the listed rules or select **User Defined** to specify the feature name yourself in the **Name** box.

System

Displays the name of the parent system.

SurfaceTrim_1 Options

Clearance

Displays the clearance value.

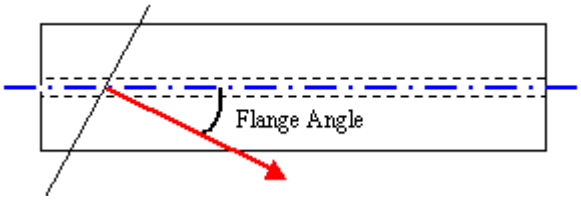
Squared End

Specifies whether the end of the member is squared or skewed when the two members are not perpendicular. Set this option to 0 if you want the member end to look like end A in the figure. Set this option to 1 if you want the member end to look like end B in the figure.



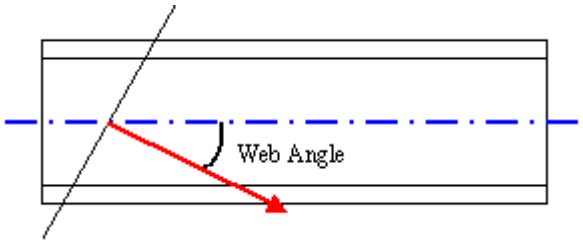
Flange Angle

Displays the angle between the normal to the cutting surface and the member axis in the horizontal plane with respect to the member (the flange plane).



Web Angle

Specifies the angle between the normal to the cutting surface and the member axis in the vertical plane with respect to the member (the web plane).



Trim End

Specifies which member end is being trimmed. Type **1** for the start. Type **2** for the end.

Web Cope 1/2 in Clearance Options

Min Cope Length Clearance

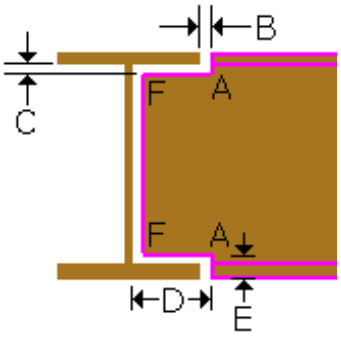
Specifies the distance between the flanges. This is dimension B in the figure.

Min Cope Depth Clearance

Specifies the distance between the web and the flange. This is dimension C in the figure.

Rounding Increment

Specifies the value by which to increment the cope dimension. For example, if you set the Increment to 2" (inches), the cope dimension is always divisible by 2 (2", 4", 6", 8", 10", and so on). Similarly, if you set the Increment to 3", the cope dimension is always divisible by 3 (3", 6", 9", 12", and so on). Setting the Increment value to 0 causes the cope to be the exact length needed to clear.



Cope Radius

Specifies the radius of the cope at the intersection of the cope length clearance and cope depth clearance. These are corners A in the figure.

Cope Radius Type

Specifies the type (**Radius** (1), **Chamfer** (2), or **None** (0)) for the cope radius.

Web Radius

Specifies the web radius of the member being coped. These are corners F in the figure.

Web Radius Type

Specifies the type (**Radius** (1), **Chamfer** (2), or **None**) (0) for the web radius.

Cope End

Used for reporting. Leave blank.

Cope Length

Specifies the cope length. This is dimension D in the figure.

Cope Depth

Specifies the cope depth. This is dimension E in the figure.

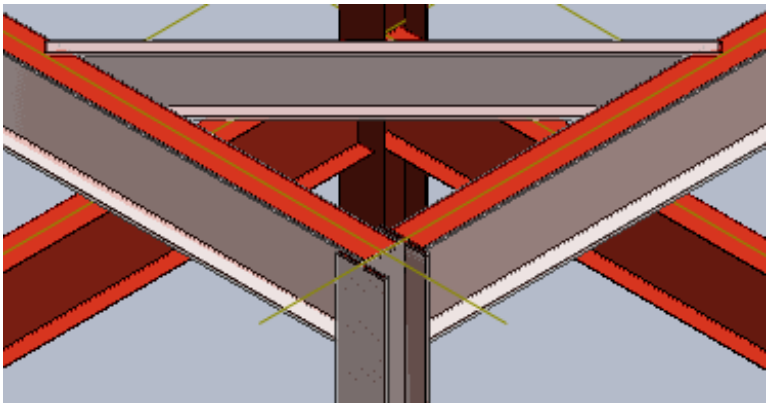
Cope Edge

Used for reporting. Leave blank.

SECTION 13

Place Fireproofing

Places fireproofing on selected members. Fireproofing is placed in the Insulation aspect, which is automatically turned on for you when insulation is placed. You can use the **Format > View** command to turn the insulation display on and off in the model as needed.



You can define fireproofing setback distances from each member end, apply multi-segmented fireproofing along a single member, and apply fireproofing to the entire model in one operation while still maintaining placement rules for different member types.

The material, grade, thickness, and rating of the fireproofing are controlled by the fireproofing specifications defined in the catalog.

NOTES

- You cannot place fireproofing on slabs, walls, plates, marine profiles, or surfaces.
- You cannot copy fireproofing directly from one member to another. However, fireproofing will copy and move with its parent member.

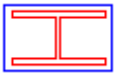
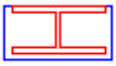
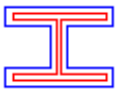
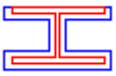
Encasement Rules

You can edit the fireproofing encasement rules in the catalog without any need for custom programming. Two encasement rules, Member Concrete and Member Fibrous, are delivered by default with the software under the **Structure > Objects > Insulation > Fireproofing** node in the **Catalog** task.

Fireproofing is specification-driven with user-modifiable encasement rules:

- Encasement rules can be applied to any cross-section, any member type, or any combination of cross-section and member type.
- Sets of encasement rules can be driven by a fireproofing specification. Therefore, you can create specifications based on a particular project or a particular client.

The standard encasement shapes (block, block-top exposed, contour, contour- top exposed, and round) are provided. You can define custom encasement shapes using the 2D Symbols editor.

Block	Block-Top Exposed	Contour	Contour-Top Exposed
			

Encasements can be placed on members using the encasement rules or can be selected manually for placement on a member.

Relationship to the Parent Member

Fireproofing must be in the same permission group as the member it is placed on. Therefore, the software automatically maintains this relationship by changing the fireproofing permission group when you change the parent member permission group. Similarly, if you change the fireproofing permission group, the software automatically changes the member permission group to match.

A similar situation is maintained for the approval status. The member's approval status cannot be "less" than the fireproofing approval status. Setting the member to a lesser approval status (from **Approved** to **In Review** for example) will cause the fireproofing to be set to the same status. Similarly, increasing the fireproofing status increases the approval status of the member.

Place Fireproofing Ribbon

Specifies the properties for the fireproofing that you are placing or editing.

Insulation Properties

Activates the **Insulation Properties** dialog box. You can use this dialog box to specify additional properties that you cannot set on the ribbon. For more information, see *Insulation Properties Dialog Box* (on page 213).

Select Members

Activated automatically by the software so that you can select the members on which you want to place fireproofing/insulation.

Finish

Click to place the fireproofing on the selected members.

Cancel

Rejects the selected object.

Accept

Confirms that the selected members are the members on which to place fireproofing. The software displays fireproofing in tentative mode.

By Rule

Allows the software to select the fireproofing to use based on the selected member and the encasement rules defined in the catalog. You can edit encasement rules by going to the Catalog task and navigating to the **Structure > Objects > Insulation > Fireproofing** node.

Condition

Specifies how you want the software to handle existing fireproofing when you try to place a

new fireproofing on the same member. Select **Retain** to keep the existing fireproofing. Select **Add** to add additional fireproofing to a member that already has fireproofing.

Insulation Specification

Select the insulation specification that you want to use. You can edit existing or define new insulation specifications in the Catalog task by navigating to the **Structure > Object Specifications > Insulation > Fireproofing** node.

Encasement

Select the encasement that you want to use from the Catalog. Available encasements depend on the insulation specification that you have selected. If you select the **By Rule** option, then the encasement option is not available because the **Encasement** item is selected by the encasement rule.

Reference 1

Select the reference point from which to cut back the fireproofing. If you select the **By Rule** option, then the **Reference 1** option is not available because the **Reference 1** value is defined in the encasement rule.

- **Part Start** - The start of the member part, which takes into account any assembly connection cutbacks or other features which may change member length.
- **Part End** - The end of the member part, which takes into account any assembly connection cutbacks or other features which may change member length.
- **Axis Start** - The start of the member part's axis, which will remain unaffected by member part features.
- **Axis End** - The end of the member part's axis, which will remain unaffected by member part features.

Distance 1

Specifies the distance between the reference 1 location and that end of the fireproofing. If you select the **By Rule** option, then this option is not available because the **Distance 1** value is defined in the encasement rule.

Reference 2

Select the reference point from which to cut back the fireproofing. If you select the **By Rule** option, then this option is not available because the **Reference 2** value is defined in the encasement rule.

- **Part Start** - The start of the member part, which takes into account any assembly connection cutbacks or other features which may change member length.
- **Part End** - The end of the member part, which takes into account any assembly connection cutbacks or other features which may change member length.
- **Axis Start** - The start of the member part's axis, which will remain unaffected by member part features.
- **Axis End** - The end of the member part's axis, which will remain unaffected by member part features.

Distance 2

Specifies the distance between the reference 2 location and that end of the fireproofing. If you select the **By Rule** option, then this option is not available because the **Distance 2**

value is defined in the encasement rule.

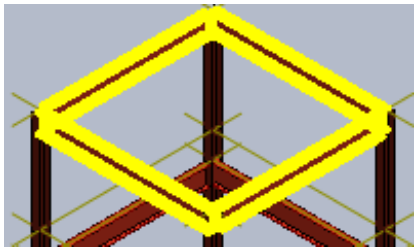
What do you want to do?

- *Place fireproofing automatically* (on page 210)
- *Place fireproofing manually* (on page 211)
- *Modify fireproofing properties* (on page 211)
- *Place two types of fireproofing on a single member* (on page 212)
- *Delete fireproofing* (on page 212)

Place fireproofing automatically

1. Click **Place Fireproofing** on the vertical toolbar.
2. Select the **By Rule** option to use the fireproofing properties defined in the encasement rules.
3. Select the member to which you want to apply the fireproofing.

   TIP You can select more than one member by dragging a selection fence around the members.

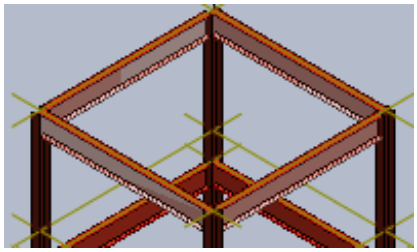
4. Set **Condition** to **Add**.

   TIP Set **Condition** to **Retain** if members in the selection set already have fireproofing and you do not want additional fireproofing to be placed on them.

5. In the **Insulation Spec** box, select the insulation specification to use.
6. Click **Accept** ✓.

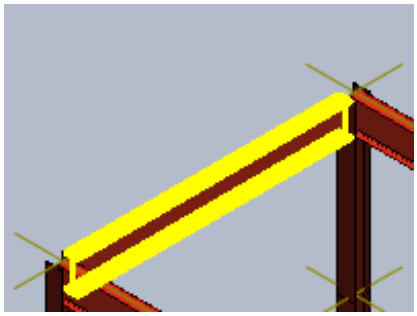
   The software tentatively displays the fireproofing using the settings that you have specified.

7. Click **Finish** to place the fireproofing.

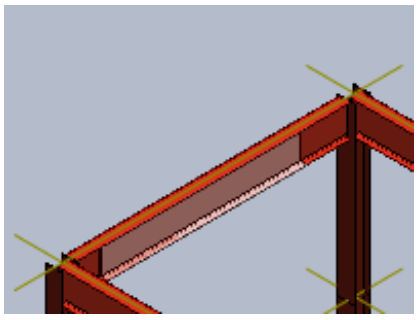
Place fireproofing manually

1. Click **Place Fireproofing** on the vertical toolbar.
2. Clear the **By Rule** option to set the fireproofing properties yourself.
3. Select the member to which you want to apply the fireproofing.



4. Set **Condition** to **Add** to create additional fireproofing on members that already have fireproofing. Set to **Retain** to skip members that already have fireproofing.
5. In the **Insulation Spec** box, select the insulation specification to use.
6. In the **Encasement** box, select the encasement to use.
7. Define any fireproofing setbacks that you need using the **Reference 1**, **Distance 1**, **Reference 2**, and **Distance 2** boxes.
8. Click **Accept** ✓.

   The software tentatively displays the fireproofing using the settings that you have specified.
9. Click **Finish** to place the fireproofing.



Modify fireproofing properties

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Insulation**.
3. Select the fireproofing to modify.
4. Edit the fireproofing properties as needed using the ribbon or properties dialog box.

Place two types of fireproofing on a single member

1. Click **Place Fireproofing** on the vertical toolbar.
2. Clear the **By Rule** option to set the fireproofing properties.
3. Select the member to which you want to apply the different types of fireproofing.
4. Set **Condition** to **Add**.
5. In the **Insulation Spec** box, select the first insulation specification to use.
6. In the **Encasement** box, select the first encasement to use.
7. Define the fireproofing setbacks that you need for this fireproofing using the **Reference 1**, **Distance 1**, **Reference 2**, and **Distance 2** boxes.
8. Click **Accept** ✓.
9. Verify the fireproofing is where you want it on the member. Adjust the setback properties if needed.
10. Click **Finish** to place the first fireproofing type.
11. Visually verify that the member is still selected (yellow highlighted). If it is not highlighted, select the member again.
12. Change the insulation specification and encasement as needed for the second type of fireproofing that you want to place on the member.
13. Define the fireproofing setbacks for the second type using the **Reference 1**, **Distance 1**, **Reference 2**, and **Distance 2** boxes.
14. Click **Accept** ✓.
15. Verify the fireproofing is where you want it on the member. Adjust the setback properties if needed to keep it from overlapping the first fireproofing that you placed.
16. Click **Finish** to place the second fireproofing type.

Delete fireproofing

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Insulation**.
3. Select the fireproofing to delete.

   TIP You can delete fireproofing from more than one member by dragging a selection fence around the members on which you want to delete the fireproofing.
4. Click **Delete** ✕.

Insulation Properties Dialog Box

Specifies the properties for the insulation that you are editing.

See Also

Occurrence Tab (Insulation Properties Dialog Box) (on page 213)
Definition Tab (Insulation Properties Dialog Box) (on page 217)
Specification Tab (Insulation Properties Dialog Box) (on page 219)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

Occurrence Tab (Insulation Properties Dialog Box)

The **Occurrence** tab displays the insulation properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one insulation, and then selected the **Properties** command, only the common properties between the selected insulation display.

When viewing properties for a single insulation type, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the Structure Reference Data Guide for more information on properties.

Standard

Name

Displays the name of the insulation. The insulation name is based on the **Name Rule** selection. If you want to type a new name for the insulation, in the **Name Rule** box, select **User Defined**, and then type a name for the insulation in the **Name** box.

Name Rule

Specifies the naming rule that you want to use to name this insulation.

- **Default Name Rule** - Names the insulation using the format: <encasement name>-<location>-<index> where <encasement name> is the insulation specification encasement name, <location> is the global workshare locationID, and <index> is a unique index number that starts at 0001. For example, Column_Contour Encased_All_Fibr-1-0003.
- **Unique Name Rule** - Names the insulation using the format: <encasement name>-<location>-<index> where <encasement name> is the insulation specification encasement name, <location> is the global workshare locationID, and <index> is a unique index number that starts at 0001. For example, Column_Contour Encased_All_Fibr-1-0003.
- **Catalog Default Name Rule** - Names the insulation using the format: <encasement name>-<location>-<index> where <encasement name> is the insulation specification encasement name, <location> is the global workshare locationID, and <index> is a unique index number that starts at 0001. For example, Column_Contour Encased_All_Fibr-1-0003.
- **User Defined** - Select this option to specify the insulation name yourself in the **Name** box.

System

Selects the system to which the insulation belongs. You can create new systems in the **Systems and Specifications** task.

By Rule

Displays **True** if the insulation was placed using an encasement rule. Displays **False** if the insulation was placed with the user selecting the insulation properties manually.

Surface Area

Specifies the measurement of the surface area of insulation.

Volume

Displays the volume of the insulation.

Reporting Requirements

Specifies whether or not this insulation is reported.

Reporting Type

Selects the reporting requirements code for the insulation. Valid codes are defined in the **Catalog** task in the **Reporting Type** select list.

Setback Distance 1

Specifies the distance between the reference 1 location and that end of the fireproofing.

Setback Distance 2

Specifies the distance between the reference 2 location and that end of the fireproofing.

Setback Reference 1

Select the reference point from which to cutback the fireproofing.

- **Part Start** - The start of the member part, which takes into account any assembly connection cutbacks or other features which may change member length.
- **Part End** - The end of the member part, which takes into account any assembly connection cutbacks or other features which may change member length.
- **Axis Start** - The start of the member part's axis, which will remain unaffected by member part features.
- **Axis End** - The end of the member part's axis, which will remain unaffected by member part features.

Setback Reference 2

Select the reference point from which to cutback the fireproofing.

- **Part Start** - The start of the member part, which takes into account any assembly connection cutbacks or other features which may change member length.
- **Part End** - The end of the member part, which takes into account any assembly connection cutbacks or other features which may change member length.
- **Axis Start** - The start of the member part's axis, which will remain unaffected by member part features.
- **Axis End** - The end of the member part's axis, which will remain unaffected by member part features.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material density multiplied by the object's solid volume. Therefore, the material of the object affects the weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

NOTE For equipment, the **Weight and CG** property **Wet Weight** is the sum of **Dry Weight** and **Water Weight**. The dry weight and water weight values are catalog properties entered on the part sheet for the equipment.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Wet CG X

Specifies the X-axis location of the wet center-of-gravity.

Wet CG Y

Specifies the Y-axis location of the wet center-of-gravity.

Wet CG Z

Specifies the Z-axis location of the wet center-of-gravity.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Surface Treatment and Coating

Exterior Coating Requirement

Specifies the coating requirement for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Exterior Coating Area

Specifies the area of the coating for the object.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

See Also

Insulation Properties Dialog Box (on page 213)

Definition Tab (Insulation Properties Dialog Box)

The Definition tab displays the insulation properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one insulation and then selected the properties command, only the common properties between the selected insulation display.

The properties that display depend on what you defined in the reference data. Refer to the Structure Reference Data Guide for more information on the properties.

Name

Defines the encasement item name. The delivered encasement items use the naming convention of Member Type_Encasement Type_Cross Section_Material. We recommend that you use a similar naming convention or adopt the delivered naming convention.

Cross Section Type

Defines the cross section type to which the encasement option applies. The available cross section types depend on the section libraries that you have included in your Catalog. Examples of valid cross section types include: 2L, L, HSSC, W, and so forth. Use an asterisk (*) to indicate that the encasement item is applicable to all cross section types.

Definition

Defines the ProgID of the encasement rule. The delivered definition is `SPSMemberEncasements.MemberEncaseDef`.

Encasement Type

Specifies the encasement type. You can add encasement types by editing the **Member Fireproofing Application Type** select list in the Catalog task.

1 Contour Fully Encased

2 Contour Flange Exposed

3 Block Fully Encased

4 Block Flange Exposed

5 Block with Voids

6 Block with Voids Flange Exposed

7 through 988 are reserved by Intergraph for future use

999 is Custom

Fireproofing Label

Defines the fireproofing label to use on drawings.

Member Type Category

Defines the member type to which the encasement item applies. Member types are defined

in the **Structural Member Type** select list in the Catalog task.

Setback Distance 1

Defines the distance between the **Setback Reference 1** location and the insulation/fireproofing.

Setback Distance 2

Defines the distance between the **Setback Reference 2** location and the insulation/fireproofing.

Setback Reference 1

Defines the location along the member from which to measure the insulation/fireproofing setback.

- **Part Start** - The start of the member part, which takes into account any assembly connection cutbacks or other features which may change member length.
- **Part End** - The end of the member part, which takes into account any assembly connection cutbacks or other features which may change member length.
- **Axis Start** - The start of the member part's axis, which will remain unaffected by member part features.
- **Axis End** - The end of the member part's axis, which will remain unaffected by member part features.

Setback Reference 2

Defines the location along the member from which to measure the insulation/fireproofing setback.

Symbol Definition

Defines the ProgID of the insulation symbol.

See Also

Insulation Properties Dialog Box (on page 213)

Specification Tab (Insulation Properties Dialog Box)

The Specification tab displays the insulation specification properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one insulation and then selected the properties command, only the common specification properties between the selected insulation display.

The properties that display depend on what you defined in the reference data. Refer to the Structure Reference Data Guide for more information on the properties.

Specification Name

Defines the insulation specification name. All insulation specification names should be unique.

Specification Description

Describes the insulation specification.

Purpose

Defines the purpose of the insulation specification. The insulation specification purpose is defined in the **StructInsulationPurpose** select list in Catalog.

Material Name

Defines the insulation material. This material is defined in Catalog in the **Structure > Materials > Properties** node.

Material Grade

Specifies the object material grade. To change the options on the list, edit the **Structure > Materials > Properties** node in Catalog.

Thickness

Defines the insulation thickness.

Encasement Rule

Defines the encasement rules that this insulation specification must use. Encasement rules are defined in Catalog in the **Structure > Objects > Insulation > Fireproofing** node or in the **StructMemberFireproofing** workbook.

Fireproofing Rating

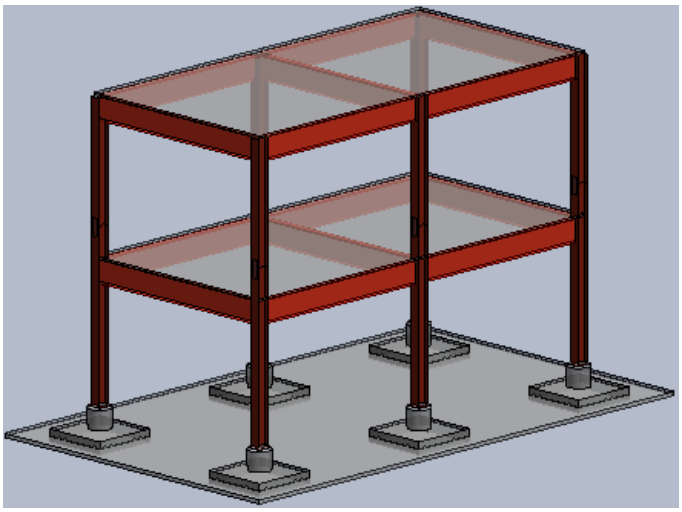
Defines the time for which the fireproofing is rated.

See Also

Insulation Properties Dialog Box (on page 213)

SECTION 14

Place Slabs



Places a slab in the model. Slabs are used to model solid surfaces, such as floors or steel grating, in your model. The software places slabs on a supporting plane, or multiple planes, that you select and between selected boundaries. The supporting plane of the slab can be an elevation plane, a grid plane, or a plane that you define during placement. You can define planes at an angle to an existing elevation or grid plane, or you can define a plane in space using three points.

The slab boundaries that you can select include grid lines, members, edges of members, edges of equipment, faces of other slabs, and many other objects in the model. When an object that is used as a boundary moves, the software automatically updates the slab to reflect the move.

You also have the option to sketch the boundaries of the slab in the model's 3D or 2D environments, or combine the selecting of boundaries for some sides of the slab and sketching the other sides.

NOTE We recommend that you place grid planes using the **Grids** task before placing slabs so that supporting planes are available for selection.

Place Slab Ribbon

Specifies the properties for the slab that you are placing.

Slab Properties

Activates the **Slab Properties** dialog box. You can use this dialog box to specify additional slab properties, such as material and material grade, which you cannot set on the ribbon. For more information, see *Slab Properties Dialog Box* (on page 234).

Define Plane

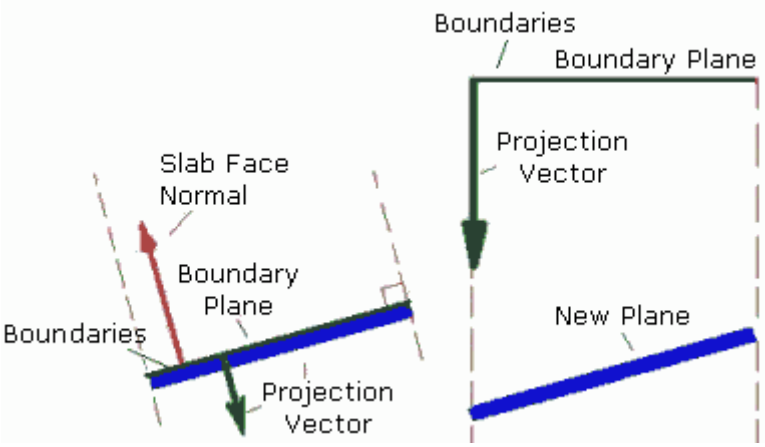
Select the support plane of the slab. You can select multiple planes to place multiple slabs at the same time. For example, if you have a multi-story building with the same floor geometry and you want the same size and type of slab on each floor, you can select the plane for each floor. You can select each floor elevation plane, and then define the boundaries on one floor. The software uses the same boundaries for each elevation level and places the slabs. After placement, the slabs are unrelated to each other and can be edited as individual entities.

Custom Sketching Plane

When the support plane is sloped (you used **3 Point Plane** or **Angle to Plane** to define the support plane), you can select a custom sketching plane on which to define the boundaries.

When placing a sloped slab, the default boundary plane is normal to the slab face (left side of the figure). The boundaries that you define are projected normal to sloped slab face.

However, if you select the **Custom Sketching Plane** option on the ribbon, the software prompts you to select a boundary plane (usually an elevation plane). The boundaries that you defined on the custom boundary plane are projected parallel to the Z-axis (right side of the figure).

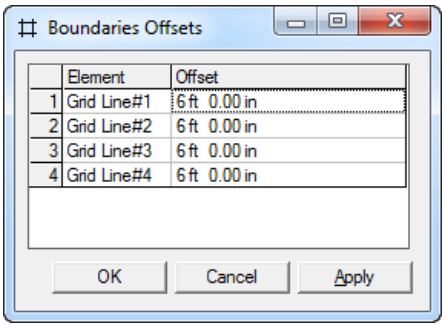


Define Boundaries

Define the outside boundaries of the slab. You can define the slab boundaries by selecting objects in the model. If you select objects in the model and those objects are moved, the software automatically resizes the slab to maintain the boundary relationship. For example, you can select the outside edge of a beam flange as a boundary along one side of a slab. Later, if the section size for the beam is changed and the flange edge moves out an inch, the software automatically moves the edge of the slab an inch to match the new location of the flange edge.

Define Boundary Offsets

Activates the **Boundary Offsets** dialog box. This dialog box allows you to define an offset distance for each boundary that you have defined. Select a row to highlight the boundary in the model. Then, type the distance from the boundary to place the edge of the slab. Type 0 to place the slab edge on the boundary. Type a negative number to offset the slab inside the boundary. Type a positive number to offset the slab outside the boundary.



Finish

Places the slab using the defined parameters.

Plane Method

Specifies how you want to define the support plane. This option is only available when you are defining the support plane.

- Select **Coincident** to specify that you want the slab placed on the support plane.
- Select **Offset from a Plane** to place the slab a specified distance from the support plane. If you choose this option, you must define the offset distance.
- Select **Angle to plane** to place the slab at a specified angle or slope to the support plane. If you choose this option, you must define an axis of rotation and the angle or slope.
- Select **Vector & Point** to specify the support plane using two points to define a vector normal to the plane and a third point to define the plane position along the vector.
- Select **3 Point Plane** to specify the support plane using three points that you specify in the model.

Offset

Specify the offset distance for the slab from the selected support plane. You can specify the offset dynamically in graphics or by typing the distance. This option is only available when **Plane Method** is set to **Offset from a Plane**. Click **Lock/Unlock** to lock the offset distance.

Angle

Specify the angle at which to place the slab relative to the support plane. You have to define the axis of rotation using two points before you can define the angle. This option is only available when **Plane Method** is set to **Angle to plane**. Click **Lock/Unlock** to lock the angle.

Slope

Specify the slope at which to place the slab relative to the support plane. You have to define the axis of rotation using two points before you can define the slope. This option is only available when **Plane Method** is set to **Angle to plane**. Click **Lock/Unlock** to lock the slope.

Select Boundaries

Select this option to select objects in the model to define the slab boundaries. This option is only available when you are defining the slab boundaries.

Add References to Sketch 2D

Allows you to select which objects in the 3-D environment to see in the sketch 2-D environment when you are drawing the boundaries of the slab. The objects that you select will display as thick blue lines in the 2-D environment. You can define relationships and dimensions to these blue-lined objects in 2-D. The software may also automatically retrieve 3-D objects to display in the 2-D environment. The automatically selected objects will display as grey lines. These grey line objects are read-only (you cannot define relationships or dimensions to them) and are there to help you visualize the model. Member center lines are also automatically displayed as dash-dot lines in the 2-D sketches. This option is only available when you are drawing the slab boundaries.

NOTE For piping and equipment objects in plant mode, **Insulation**, **Equipment Hole**, **Maintenance**, and **Operation** aspects displaying in the 3D graphic view also display in the Sketch 2D environment. The aspect geometry displays as dashed lines. You can define dimensions, but not relationships, to these aspects in 2D.

NOTE For slabs and walls defined in the sketch 2-D environment, Smart 3D removes any constraints external to the defining group when you:

- Copy a slab or wall. The original slab or wall will still have the constraints; however, the newly created copy of the slab or wall will not have the external constraints.
- Move a slab or wall. All external constraints are removed from the slab or wall.
- Rotate a slab or wall. All external constraints are removed from the slab or wall.

The defining group is the lines (or other shapes) that you place in the sketch 2-D environment that defines the actual slab or wall. An example of an external to the group constraint might be an edge of a Slab XZY offset from the edge of Slab 123. The constraint is between two different slabs (defined by two different groups) so it is removed. An example of an internal to the group constraint that is not removed might be one slab side constrained to be parallel to the opposite side. In this case both sides belong to the same slab, and hence the same group, and the constraint is therefore not removed.

 **Draw**

Opens the sketch 2-D environment in which you can draw the boundaries of the slab. You can draw multiple slabs at a time in the 2-D environment. This option is only available when you are defining the slab boundaries.

 **Select Patches to Compose the Slab**

If the boundaries that you selected can form the boundary for the slab in more than one way, then you have defined an ambiguous boundary. The software prompts you to select one or more bounded areas to clarify the required slab boundary. For more information, see *Solve Ambiguous Boundaries* (on page 232).

 **Sketch 3D**

Select this option to sketch the boundaries of the slab in the 3-D environment.

 **Cancel**

Clears all selected planes or boundary objects.

 **Accept**

Accepts all selected planes or boundary objects.

System

Select the parent system for the slab that you are placing. Systems are defined using the Systems and Specifications task.

Type

Select the type of slab to place, such as a slab on grade or composite deck. The properties change depending on the slab type that you select. You can define a custom slab type in the **StructSlabGeneral.xls** workbook in the reference data. Refer to the Structure Reference Data Guide for more information.

Composition

Select a slab composition from the list. Available compositions depend on the type of slab being placed. You can define a custom composition using the **StructSlabGeneral.xls** and **StructSlabLayer.xls** workbooks in the reference data. Refer to the Structure Reference Data Guide for more information.

Name

Displays the name of the slab that you are placing.

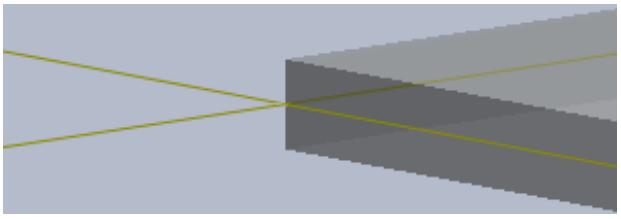
Priority

Specifies the priority assigned to the object system. Smart 3D uses the priority to group objects. To change the options on the list, edit the **Structural Member Priority** select list in Catalog.

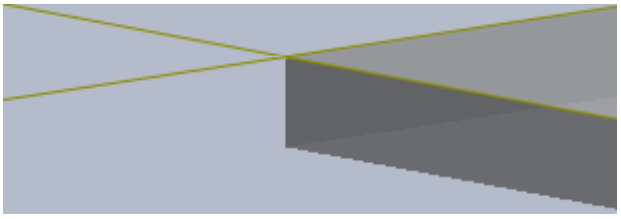
Face Position

Specifies the way in which the slab is placed in the model with respect to the support plane. To change the options on the list, edit the **Slab Face Position** select list in Catalog.

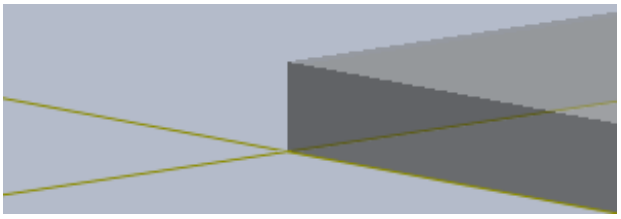
Center indicates that the slab is placed with respect to its centroidal axis.



Top indicates that the slab is placed with respect to its top face.



Bottom indicates that the slab is placed with respect to its bottom face.



Total Thickness

Type the thickness of the slab. The default thickness is based on the selected slab type and composition.

NOTE If the slab has more than one layer, then the **Total Thickness** box is read-only. To edit the slab thickness, use the **Layer** tab of the **Slab Properties** dialog box. For more information, see *Edit Slab Properties* (on page 234).

Slab Assembly Connections

Slab assembly connections are similar to member assembly connections, but define the trimming and edges of slabs. Slab assembly connections are placed automatically when the slab is created.

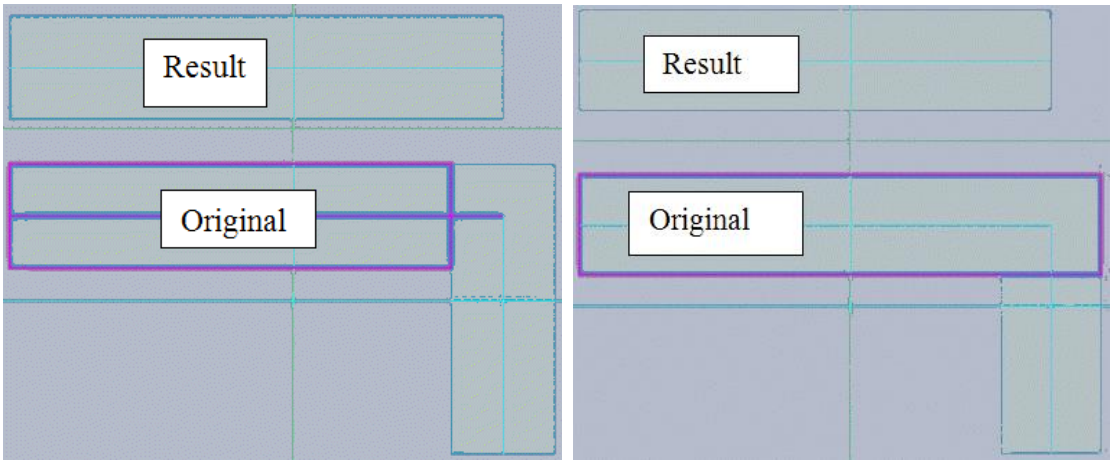
The slab assembly connections are:

- The Slab by Member Boundary assembly connection is placed by the software when you select a member as a slab boundary.

- The Slab Free Edge Assembly connection is placed by the software when you select a grid line as a slab boundary.

If a slab's bounding object (for example a member, a wall, or another slab) is deleted, then that deleted object is replaced as a boundary by the slab assembly connection which is generated from that deleted object. This behavior keeps the slab from going to the **To Do List** as missing a boundary.

In addition, if you copy, move, or mirror a wall (including Model Data Reuse operations) and an object which was connected to the wall is not in the copy set and the **Delete optional** option is selected, the software changes the wall assembly connection for the missing object to be a Wall Free Edge Assembly connection. This prevents the wall from going to the **To Do List** with a "Wall is missing a boundary Select a valid boundary" message. Be aware that copying or moving a wall with the **Delete optional** option selected can cause the geometry of the wall to extend or shorten.



★ **IMPORTANT** This geometry change impacts the use of the **Validate Model Data Reuse Results** command because the geometry of the wall is different between the original wall and the new wall.

Locating Slab Assembly Connections

Assembly connections do not display in the model. However, if you set the **Locate Filter** to **Slab Assembly Connections**, you can locate and select assembly connections for review and editing. Assembly connections are located at the edges of slabs.

When you select an assembly connection, the software displays the assembly connection type in the ribbon. Select the **Edit > Properties** command to edit the assembly connection properties.

What do you want to do?

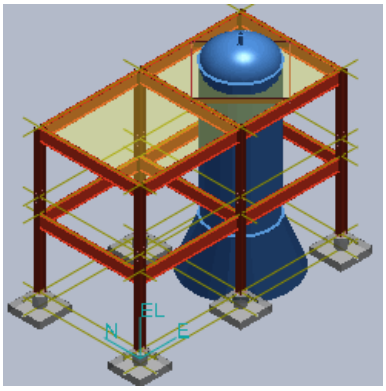
- *Place a slab by selecting boundary objects* (on page 226)
- *Place a slab by drawing 2-D boundaries* (on page 228)
- *Place a slab by drawing 3-D boundaries* (on page 230)
- *Place a slab by selecting objects and drawing 2-D boundaries* (on page 230)
- *Place a slab by selecting objects and drawing 3-D boundaries* (on page 231)

- *Solve ambiguous boundaries* (on page 232)
- *Edit slab object boundaries* (on page 232)
- *Edit 2-D drawn slab boundaries* (on page 232)
- *Edit 3-D drawn slab boundaries* (on page 233)
- *Edit slab properties* (on page 234)
- *Delete a Slab* (on page 234)

Place a slab by selecting boundary objects

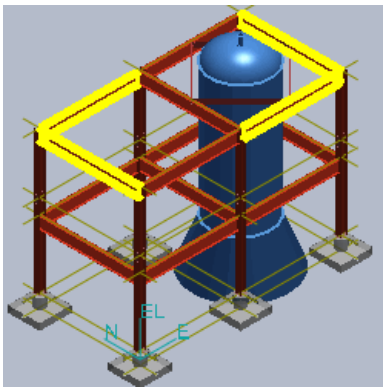
1. Click **Place Slab** on the vertical toolbar.
2. Set the parent system for the slab, the slab type and composition, and face position.
3. Define a plane for the slab.

   Define Plane as Coincident (on page 242)
   Define Plane using Offset from Plane (on page 242)
   Define Plane using Angle to Plane (on page 242)
   Define Plane using Vector and Point (on page 243)
   Define Plane using Three Points (on page 243)



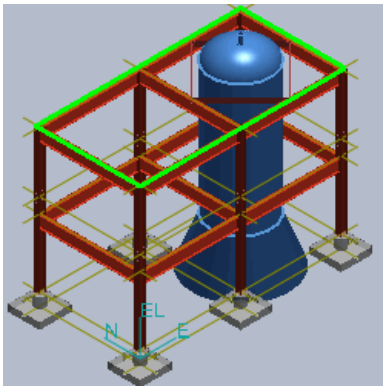
4. Click **Accept** ✓.
5. Click **Select Boundaries**.

6. Select objects in the model to define the boundaries of the slab. You can select members, grid lines, edges of equipment, edges of other slabs, or most anything else in the model to define the slab boundaries. The selected objects must roughly define a close shape.

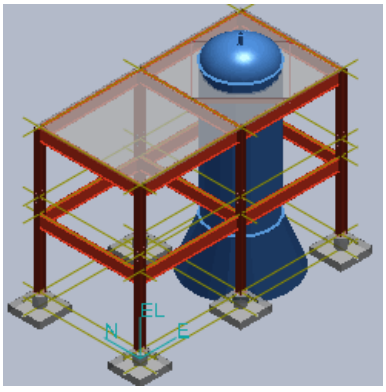


7. Click **Accept** ✓.

   The software displays the slab in dynamics.



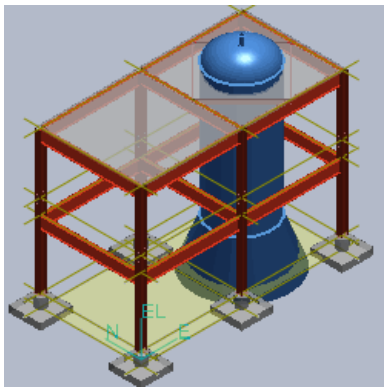
8. Click **Finish**.



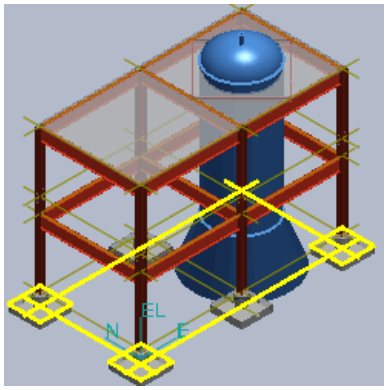
Place a slab by drawing 2-D boundaries

1. Click **Place Slab** on the vertical toolbar.
2. Define a plane for the slab.

   Define Plane as Coincident (on page 242)
   Define Plane using Offset from Plane (on page 242)
   Define Plane using Angle to Plane (on page 242)
   Define Plane using Vector and Point (on page 243)
   Define Plane using Three Points (on page 243)

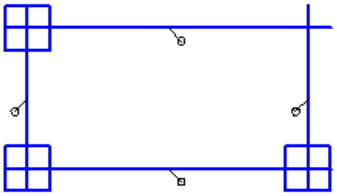


3. Click **Accept** ✓.
4. Click **Add References to Sketch 2D**.
5. Select objects in the model that you want to see when drawing the slab boundaries in the 2-D environment.

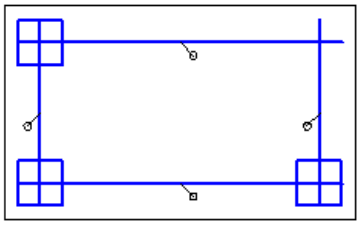


6. Click **Draw**.

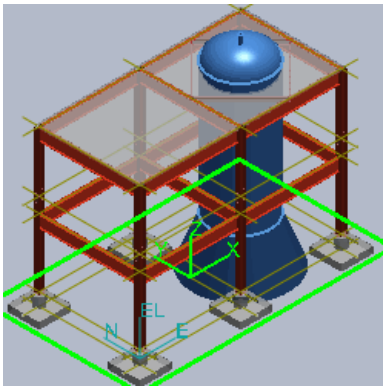
   TIP The 2D sketch orientation is based on the active coordinate system as defined in the **Pin Point** command.



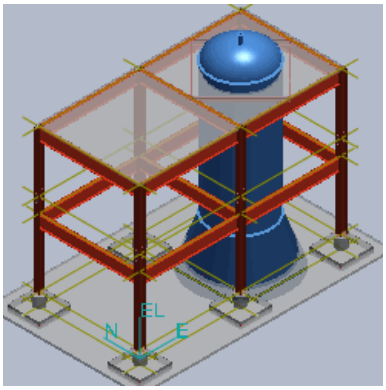
7. In the 2-D environment, draw the boundaries of the slab. The boundaries that you draw must be closed shape.



8. In the 2-D environment, click **Close**.
9. Click **Accept**.



10. Set the slab system, type, priority, face position, and other properties.
11. Click **Finish**.



NOTE For slabs and walls defined in the sketch 2-D environment, Smart 3D removes any constraints external to the defining group when you:

- Copy a slab or wall. The original slab or wall will still have the constraints; however, the newly created copy of the slab or wall will not have the external constraints.
- Move a slab or wall. All external constraints are removed from the slab or wall.
- Rotate a slab or wall. All external constraints are removed from the slab or wall.

The defining group is the lines (or other shapes) that you place in the sketch 2-D environment that defines the actual slab or wall. An example of an external to the group

constraint might be an edge of a Slab XZY offset from the edge of Slab 123. The constraint is between two different slabs (defined by two different groups) so it is removed. An example of an internal to the group constraint that is not removed might be one slab side constrained to be parallel to the opposite side. In this case both sides belong to the same slab, and hence the same group, and the constraint is therefore not removed.

Place a slab by drawing 3-D boundaries

1. Click **Place Slab** on the vertical toolbar.
2. Define a plane for the slab.

   Define Plane as Coincident (on page 242)
   Define Plane using Offset from Plane (on page 242)
   Define Plane using Angle to Plane (on page 242)
   Define Plane using Vector and Point (on page 243)
   Define Plane using Three Points (on page 243)
3. Click **Accept** ✓.
4. Click **Sketch 3D** 3D.
5. Sketching a path that defines the outside boundary of the slab.

   Define a Path (on page 501)
6. Click **Finish**.
7. Click **Accept** ✓.
8. Set the slab system, type, priority, face position, and other properties.
9. Click **Finish**.

Place a slab by selecting objects and drawing 2-D boundaries

1. Click **Place Slab** on the vertical toolbar.
2. Define a plane for the slab.

   Define Plane as Coincident (on page 242)
   Define Plane using Offset from Plane (on page 242)
   Define Plane using Angle to Plane (on page 242)
   Define Plane using Vector and Point (on page 243)
   Define Plane using Three Points (on page 243)
3. Click **Accept** ✓.
4. Click **Select Boundaries**.
5. Select objects in the model to define part of the slab boundaries. You can select members, grid lines, edges of equipment, edges of other slabs, or most anything else in the model to define the slab boundaries.
6. Click **Add References to Sketch 2D**.

7. Select objects in the model that you want to see when drawing the slab boundaries in the 2-D environment.
8. Click **Draw** .
9. In the 2-D environment, draw the remaining boundaries of the slab. The object boundaries that you selected are displayed in black. The objects that you selected for sketch landmarks display in blue. The boundaries that you selected and the boundaries that you draw must form closed shape. You can draw the boundaries for multiple slabs at the same time in the 2-D environment.
10. In the 2-D environment, click **Close**.
11. Click **Accept** .
12. Set the slab system, type, priority, face position, and other properties.
13. Click **Finish**.

Place a slab by selecting objects and drawing 3-D boundaries

1. Click **Place Slab** on the vertical toolbar.
2. Define a plane for the slab.

   Define Plane as Coincident (on page 242)
   Define Plane using Offset from Plane (on page 242)
   Define Plane using Angle to Plane (on page 242)
   Define Plane using Vector and Point (on page 243)
   Define Plane using Three Points (on page 243)
3. Click **Accept** .
4. Click **Select Boundaries** .
5. Select objects in the model to define part of the slab boundaries. You can select members, grid lines, edges of equipment, edges of other slabs, or most anything else in the model to define the slab boundaries.
6. Click **Sketch 3D** .
7. Sketching a path that defines the remaining part of the slab boundary.
8. *Define a Path* (on page 501)
9. Click **Finish**.
10. Click **Accept** .
11. Set the slab system, type, priority, face position, and other properties.
12. Click **Finish**.

Solve ambiguous boundaries

The command switches to the solve ambiguity mode automatically if ambiguous boundaries exist after clicking **Accept** or **Finish**. All possible bounded areas appear in the graphic view outlined in green.

1. Move the pointer over a bounded area, and then click to select that bounded area. The selected area highlights in yellow.
2. Continue to select bounded areas until you have defined the entire bounded area that you want. A correct solution to ambiguity meets these conditions:
   - Each bounded area must have a common edge with at least one other bounded area.
   - Each boundary must be used by at least one bounded area.

NOTE To remove a bounded area, select the area again.

Edit slab object boundaries

1. Click **Select** on the vertical toolbar.
2. In the **Locate Filter**, select **Slabs**.
3. Select the slab to edit.
4. On the ribbon, click **Define Boundaries**.
5. Redefine the boundaries by clicking existing boundaries to clear them and by clicking new objects to use as the new boundaries. For more information, see *Solve Ambiguous Boundaries* (on page 232).
6. Click **Accept**.
7. Click **Finish**.

NOTE You can only edit the exterior boundaries of a slab using the **Place Slab** command. Openings in the slab should be edited with the **Place Opening** command.

Edit 2-D drawn slab boundaries

1. Click **Select** on the vertical toolbar.
2. In the **Locate Filter**, select **Slabs**.
3. Select the slab to edit.
4. On the ribbon, click **Define Boundaries**.
5. On the ribbon, click **Draw**.
6. In the 2-D environment, edit the boundary as needed by adding or removing graphic segments. The new segments need to be added or removed to/from an existing group depending on the number of SlabFreeEdgeAsm connections needed. In most of cases, there is no need to use the group modify command for slabs. Be sure to use the **Bottom Up**

option when selecting segments. When you are finished, make sure the boundary is a closed shape.

a. Addition of elements - The newly added elements are part of a new group (even if connected to another element belonging to another group). This operation creates an additional assembly connection for the new created group with the new elements. Using the group command serves to create one assembly connection.

b. Inserting an element in the middle of a chain - Inserting an element creates a new assembly connection. Using the group modify command depends if you need a single or several assembly connections.

c. Removing an element - You should not ungroup elements when dependent objects are attached to the boundaries. Removing an element from a boundary can be done by deleting the graphic element. Removing an element from a group (to create a different assembly connection) can be done by using the group modify command.

7. Click **Add or remove graphic from a group** on the **Change** toolbar.
8. Click **Select Graphic Group** on the ribbon.
9. Select the original slab boundary.
10. Click **Select elements to add to group or remove from it** on the ribbon.
11. Select the new slab boundary segments that you just placed.
12. Click **Add** on the ribbon.
13. In the 2-D environment, click **Close**.
14. Click **Accept**.
15. Click **Finish**.

NOTE You can only edit the exterior boundaries of a slab using the **Place Slab** command. Openings in the slab should be edited with the **Place Opening** command.

Edit 3-D drawn slab boundaries

1. Click **Select** on the vertical toolbar.
2. In the **Locate Filter**, select **Slabs**.
3. Select the slab to edit.
4. On the ribbon, click **Define Boundaries**.
5. On the ribbon, click **Sketch 3D**.
6. Edit the path.

   Modify a Straight Segment in a Path (on page 504)
   Modify an Arc in a Path (on page 505)
   Modify a Turn in a Path (on page 505)
   Move Segments of a Path (on page 503)
7. Click **Finish**.
8. Click **Accept**.

9. Click **Finish**.

NOTE You can only edit the exterior boundaries of a slab using the **Place Slab** command. Openings in the slab should be edited with the **Place Opening** command.

Edit slab properties

1. Click **Select** on the vertical toolbar.
2. Select **Slabs** in the locate filter.
3. Select the slab to edit.
4. Click **Edit > Properties**.
5. Edit the slab properties as needed.

Delete a slab

1. Click **Select** on the vertical toolbar.
2. Select **Slabs** in the **Locate Filter**.
3. Select the slab to delete.
4. Click **Delete**.

Slab Properties Dialog Box

Specifies the properties for the slab that you are editing. You can define the default values in the **StructSlabGeneral.xls** and **StructSlabLayer.xls** workbooks.

See Also

General Tab (Slab Properties Dialog Box) (on page 234)
Layer Tab (Slab Properties Dialog Box) (on page 241)
Edit Slab Properties (on page 234)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

General Tab (Slab Properties Dialog Box)

The **General** tab displays the slab properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one slab and then selected the properties command, only the common properties between the selected slabs display.

When viewing properties for a single slab, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on properties. You can access this document using the **Help > Printable Guides** command in the software.

Category

Select the properties that you want to view for the slab. Slab properties are divided into several different categories: Standard, Dimensions, Weight and CG, Fabrication and Construction, Surface Treatments and Coatings, and Responsibility. You select the category to define values for by using the **Category** option.

Standard

The standard properties that display depend on the reference data for the slab. Because slab reference data is fully customizable, only the common properties are documented.

Parent System

Select the system to which the slab belongs.

Name

Displays the name of the slab. The slab name is based on the **Naming Rule** selection. If you want to type a new name for the slab, in the **Naming Rule** box, select **User Defined**, and then type a name for the slab in the **Name** box.

Continuity Priority Number

Specify the continuity priority. This priority is used to select which slab is split when two slabs intersect, but both have **Intercostal** for the **Continuity Type**. Slabs with a lower continuity priority (1, 2, 3, for example) will split slabs with a higher continuity priority (7, 8, 9, for example).

Continuity Type

Defines how the slab should react when it intersects another slab (your automatic splitting preference). Select **Continuous** to indicate that the slab should split the other slab. Select **Intercostal** to indicate that the slab should be split by the other slab. You cannot split slabs that have a **Continuity Type** setting of **Continuous**.

Naming Rule

Specify the naming rule that you want to use to name this slab.

Default Name Rule - Names the slab using the format "Slab-<location>-<index>" where <location> is the global workshare location ID and <index> is a unique index number that starts at 0001. For example, Slab-1-0043.

Unique Name Rule - Names the slab using the format "<system>-Slab-<location>-<index>" where <system> is the name of the parent system to which the slab belongs, <location> is the global workshare location ID and <index> is a unique index number that starts at 0001. For example, Structure System-Slab-1-0043.

User Defined - Select to specify the slab name yourself in the **Name** box.

Type

Select the slab type.

Composition

Specifies the composition to use. You can see details of the selected composition by looking at the *Layer Tab (Slab Properties Dialog Box)* (on page 241).

Priority

Specifies the priority assigned to the object system. Smart 3D uses the priority to group objects. To change the options on the list, edit the **Structural Member Priority** select list in

Catalog.

Reporting Requirement

Specify whether or not this slab is reported.

Reporting Type

Select the reporting requirements code for the slab. Valid selections are defined in the **Catalog** task in the **Reporting Type** select list.

Piece Mark

Specifies the piece mark of the slab. Piece marks are mainly used when importing and exporting slabs through CIMsteel to other software packages. If a piece mark is displayed here, more than likely it is the identity of the manufactured part from the other software package.

Assembly Mark

Specifies the assembly mark of the member. If the member was imported through CIMsteel, the assembly mark is usually the identity of the manufactured assembly to which this member belongs.

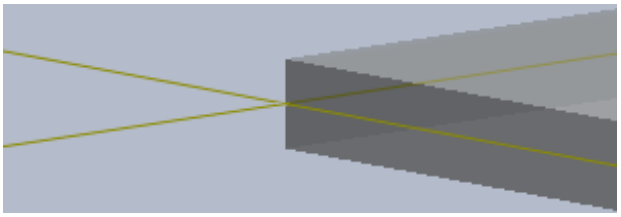
Fire Rating

Specifies the fire rating time for the slab.

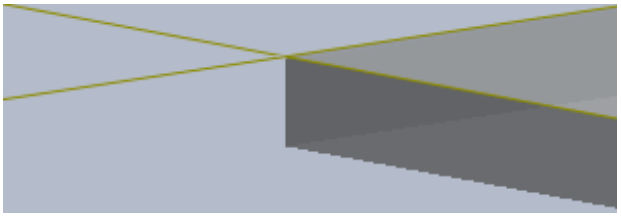
Face Position

Specifies the way in which the slab is placed in the model with respect to the support plane. To change the options on the list, edit the **Slab Face Position** select list in Catalog.

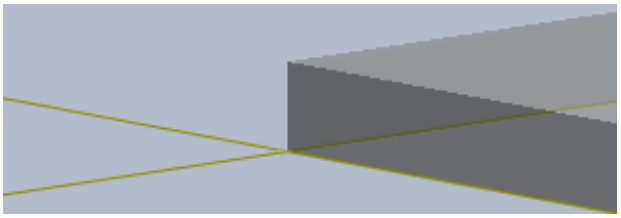
Center indicates that the slab is placed with respect to its centroidal axis.



Top indicates that the slab is placed with respect to its top face.



Bottom indicates that the slab is placed with respect to its bottom face.



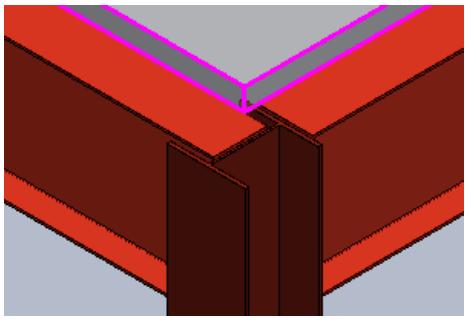
Normal Offset

Displays the offset distance for the slab from the selected support plane.

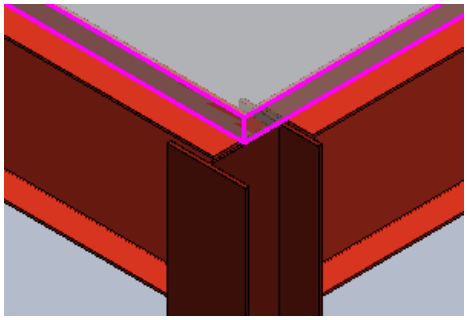
Boundary Offset Reference

Indicates which part of the member the slab should use as the boundary when a member is used as a boundary for a slab. To change the options on the list, edit the **Slab Boundary Reference** select list in Catalog.

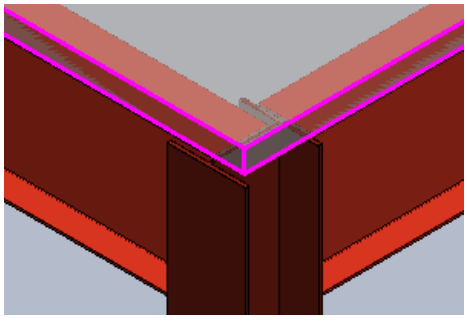
Inner port-face of a bounding member indicates that the slab stops at the inner-most plane of the bounding member.



Centerline (axis) of a bounding member indicates that the slab stops at the member-line axis of the bounding member. In the picture below, the beams were placed using cardinal point 8, which means the member-line axis is in the top center of the beams.



Outer port-face of a bounding member indicates that the slab stops at the out-most plane of the bounding member.



Boundary Offset

Specifies the boundary offsets for the slab. Enter 0 for the slab edge to stop at the boundary. Enter a positive distance to offset the slab edge outside of the boundary. Enter a negative distance to offset the slab edge inside the boundary.

Angle

Displays the angle at which the slab is placed from the plane or reference plane.

Slope

Specifies the slope of the slab relative to the reference plane that was selected during placement.

Thickening Direction

Specifies the thickening direction. To change the options on the list, edit the **Slab Reference Direction** select list in Catalog.

Boundaries Projection Direction

Specifies the boundaries projection direction. To change the options on the list, edit the **Slab Reference Direction** select list in Catalog.

Dimensions

Total Volume

Specifies the volume of the slab or wall (length × width × height).

Net Volume

Specifies the volume of the wall minus the volumes of all openings in the wall.

Total Surface

Specifies the surface area of the slab or wall (top surface plus the bottom surface plus the side surfaces × length).

Net Surface

Specifies the surface area of the slab or wall minus the surface area of all the openings in the slab or wall.

Total Projected Area

Specifies the area of the slab (length x width).

Net Projected Area

Displays the area of the slab minus the area of all the openings in the slab.

Number of Openings

Displays the number of openings in the slab.

Angle

Displays the angle at which the slab is placed from the plane or reference plane.

Low Point

Displays the lowest elevation of the slab. The elevation of the bounding objects is not considered.

High Point

Specifies the highest elevation of the slab. Smart 3D does not consider the elevation of the bounding objects.

Bottom Face

Specifies the lowest elevation of the slab or the slab boundary objects. For example, if the slab is bounded by four structural members, then the software displays the lowest elevation of either the slab or the members, whichever is lower.

Top Face

Displays the highest elevation of the slab and the slab boundary objects. For example, if the slab is bounded by four structural members that extend above the slab, the top elevation of the members is displayed.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material density multiplied by the object's solid volume. Therefore, the material of the object affects the weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight

Specifies the dry weight of the object. For slabs the weight is determined by using the volume of the slab multiplied by the density of the material.

Wet Weight

Specifies the wet weight of the object. For slabs, the weight is determined by using the volume of the slab multiplied by the density of the material.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Wet CG X

Specifies the X-axis location of the wet center-of-gravity.

Wet CG Y

Specifies the Y-axis location of the wet center-of-gravity.

Wet CG Z

Specifies the Z-axis location of the wet center-of-gravity.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Surface Treatment and Coating

Exterior Coating Requirement

Specifies the coating requirement for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Exterior Coating Area

Specifies the area of the coating for the object.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit

the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

See Also

Slab Properties Dialog Box (on page 234)

Layer Tab (Slab Properties Dialog Box)

Displays the layers for the selected slab type on the **General** tab. Layers are defined in the **StructSlabLayer.xls** workbook. Refer to the *Structure Reference Data Guide* for more information about this workbook. You can access this document using the **Help > Printable Guides** command in the software.

Composition

Select the composition that you want to use for the slab type. The layers in the select composition appear.

Total Thickness

Displays the total thickness of all the layers in the slab.

Layer

Select the layer of the composition to view or edit properties for.

Property

Displays the defined properties for the active layer.

Cross Section Symbol

Select the cross section symbol to use for the slab. This option is not currently used by the software.

Value

Specify a value for the layer property.

See Also

Slab Properties Dialog Box (on page 234)

Understanding Plane Methods

A common activity while working with slabs is defining planes. These planes are used to define the location of the slab or as a sketching plane for the slab boundaries. Usually, but not necessarily, these planes are based from an existing object or from a grid plane created using the **Grids** task. There are five methods for defining planes:



Coincident

Defines a plane coincident to another plane. For more information, see *Define Plane as*

Coincident (on page 242).

Offset from Plane

Defines a plane at a specified offset distance from another plane. For more information, see *Define Plane using Offset from Plane* (on page 242).

Angle from Plane

Defines a plane at a specified angle or slope to another plane. You must define an axis of rotation and the angle or slope. For more information, see *Define Plane using Angle to Plane* (on page 242).

Plane by Point and Vector

Defines a plane using a vector normal to the plane being defined and a third point to define the plane position along the vector. For more information, see *Define Plane using Vector and Point* (on page 243).

Plane by Three Points

Defines a plane using three points that you identify in the model. For more information, see *Define Plane using Three Points* (on page 243).

Define Plane as Coincident

1. Select **Coincident** .
2. In the model or **Workspace Explorer**, select the plane to place the new slab coincident with.

Define Plane using Offset from Plane

1. Select **Offset from a Plane** .
2. In the model or **Workspace Explorer**, select the base plane to offset the new slab from.
3. In the **Offset** box, type the offset distance from the plane.

   TIP You can also use the **Step** box on the **PinPoint** ribbon. Enter a value, and then move the pointer in the graphics view. The offset follows the pointer, incrementing by the step value that you specified.

Define Plane using Angle to Plane

1. Select **Angle from Plane** .
2. In the model or **Workspace Explorer**, select the base plane from which to angle the slab that you are defining.
3. Select a rotation axis.

   TIPS

   - The rotation axis can be a grid line or a member edge.
   - The rotation axis must be parallel to the surface of the plane selected in step 2.
4. In the **Angle** box, specify the rotation angle of the slab relative to the selected base plane.

Define Plane using Vector and Point

1. Select **Vector & Point** .
2. Define the first vector point.
3. Define the second vector point.
4. Specify the plane location normal to the vector at which to place the slab.

Define Plane using Three Points

1. Select **Plane by Three Points** .
2. Specify the first point that defines the plane.
3. Specify the second point that defines the plane.
4. Specify the third point that defines the plane.

SECTION 15

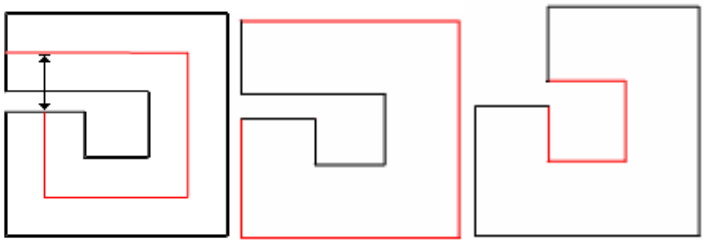
Place Walls

Places curved and straight walls in the model. Use this command to place parapets, retaining walls, bearing walls, nonbearing walls, and foundation walls. Like slabs, you can cut openings in walls using the *Place Opening* (on page 367). You can use the *Place Equipment* (on page 278) to place doors and windows in walls.

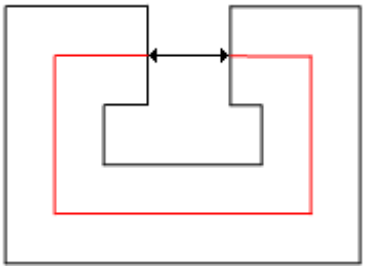
The software places walls on a supporting plane that you select. The supporting plane of the wall can be an elevation plane, a grid plane, or a plane that you define during placement.

When placing walls, you define a path that the wall is to follow. You can define this path in the 3-D environment using the same path commands that you use to define handrails paths, or you can define this path in the 2-D environment using lines connected and grouped together to define the path. When you need to modify the walls that you have placed, keep these things in mind:

- All walls paths created in the 2-D environment should be modified in the 2-D environment unless you are modifying the entire area with a rotate or move, in which case you should do the modification in the 3-D environment. For example, you placed a wall using a 2-D drawn path. To move, rotate, or add on to that individual wall, you would go back to the 2-D environment. However, if you were going to rotate the wall along with everything else in the general area, you would do the rotate all the objects in the 3-D environment.
- When moving or rotating a wall path in the 2-D environment, you might have to delete some constraints before you can move or rotate. You need to display the relationships layer to delete them if needed.
- When a 2-D sketched wall is selected in the 3-D environment, all the other walls that were created during that same 2-D session are also selected. If you move or rotate one wall, all the walls from the session will move or rotate also. To move or rotate an individual wall, you need to either place each wall individually in the 2-D environment (coming back out to the 3-D environment using Finish) or you need to go into the 2-D environment and perform the move or rotate on the individual wall in the 2-D environment.
- When sketching a wall that closes upon itself, you must make sure that the start and end points are *either* coincident or are no closer than 1 cm + 0.05 × total thickness of the wall + 0.2 meters.



If the start and end points are aligned on the same parallel vector, then the minimum distance is 1 cm.



- If you import grid lines into the 2-D environment as references and then sketch a wall path on top of the grid lines, the software will create horizontal or vertical constraints to the grid lines.

You can select multiple wall parts or wall systems as a group and edit properties either in the ribbon bar or in the property pages. The limitations when editing multiple walls are that you cannot modify support, path or boundaries values.

Place Wall Ribbon

Specifies the properties for the wall(s) that you are placing or editing. If you select multiple walls to edit, the values that are the same for the selected walls are displayed. If a value is *different* between the selected walls, the value is not shown.

IMPORTANT If you have multiple walls selected and you enter a new value in the ribbon, it is applied to ALL selected walls.

Wall Properties

Activates the **Wall Properties** dialog box. You can use this dialog box to specify additional wall properties, such as material and material grade, which you cannot set on the ribbon. For more information, see *Wall System Properties Dialog Box* (on page 265).

NOTE **Sketching Plane**, **Select a Path** and **Select Boundaries** are unavailable and cannot be edited when multiple walls are selected.

Sketching Plane

Select the support plane for the bottom of the wall.

Select a Path

Select the **Sketch 2D** or **Sketch 3D** option.

Select Boundaries

Select additional graphic objects to be boundaries for the wall. You can select members, other walls, grid planes, and surfaces. An example of when to use this option would be when placing a wall under a pitched roof.

Finish

Places the wall using the defined parameters.

Cancel

Clears all selected planes or boundary objects.

Accept

Accepts all selected planes or boundary objects.

Sketch 2D

Opens the sketch 2-D environment in which you can draw the path of the wall. You can draw more than one wall at a time in the 2-D environment. However, if you draw more than one wall path in the 2-D environment, the walls will move as a single unit if you move any of the walls regardless of whether or not the walls are connected to each other. If you will need to move any of the walls individually, you must draw that wall's path in 2-D environment by itself.

NOTE For slabs and walls defined in the sketch 2-D environment, Smart 3D removes any constraints external to the defining group when you:

- Copy a slab or wall. The original slab or wall will still have the constraints; however, the newly created copy of the slab or wall will not have the external constraints.
- Move a slab or wall. All external constraints are removed from the slab or wall.
- Rotate a slab or wall. All external constraints are removed from the slab or wall.

The defining group is the lines (or other shapes) that you place in the sketch 2-D environment that defines the actual slab or wall. An example of an external to the group constraint might be an edge of a Slab XZY offset from the edge of Slab 123. The constraint is between two different slabs (defined by two different groups) so it is removed. An example of an internal to the group constraint that is not removed might be one slab side constrained to be parallel to the opposite side. In this case both sides belong to the same slab, and hence the same group, and the constraint is therefore not removed.

Sketch 3D

Activates the Create Path Ribbon with which you define the wall path. Select this option to sketch the path of the wall in the 3-D environment.

System

Select the parent system for the wall that you are placing. Systems are defined using the Systems and Specifications task.

Type

Select the type of wall to place. The properties change depending on the wall type that you select.

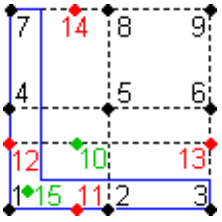
Composition

Select a wall composition from the list. Available compositions depend on the type of wall being placed.

Name

Displays the name of the wall that you are placing.

Position



Specify the relative position of the wall to the path. Fifteen positions are available. The location of positions 10 (center-of-gravity) and 15 (shear center) depend on the wall's section shape. The local z- axis of the wall and the center-of-gravity point of the section define positions 11 and 14. The local y-axis of the wall and the center-of-gravity point of the section

define positions 12 and 13.

Total Thickness

Type the thickness of the wall. The default thickness is based on the selected wall type and composition.

NOTE If the wall has more than one layer, then the **Total Thickness** box is read-only. To edit the wall thickness, use the **Layer** tab of the **Wall Properties** dialog box. For more information, see *Edit Wall Properties* (on page 264).

Maximum Height

Enter the maximum height for the wall.

Reflect

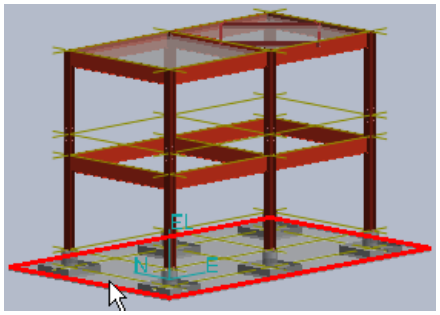
Reflects or mirrors the cross-section about the wall's path. This parameter affects both symmetric and asymmetric sections.

What do you want to do?

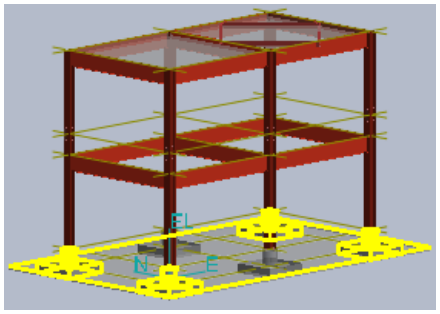
- *Place a wall by 2-D sketch* (on page 248)
- *Add end segment to existing wall using 2-D sketch* (on page 250)
- *Place split walls by 2-D sketch* (on page 251)
- *Add fillet corner to existing wall using 2-D sketch* (on page 252)
- *Add middle segments to existing wall using 2-D sketch* (on page 254)
- *Modify wall length using 2-D sketch* (on page 257)
- *Create gap in existing wall using 2-D sketch* (on page 259)
- *Rotate an existing wall using 2-D sketch* (on page 260)
- *Place a wall by 3-D sketch* (on page 261)
- *Place split walls by 3-D sketch* (on page 262)
- *Split an existing wall using 3-D sketch* (on page 263)
- *Add end segment to existing wall using 3-D sketch* (on page 263)
- *Modify wall cross-section* (on page 264)
- *Edit wall properties* (on page 264)
- *Edit multiple walls* (on page 264)
- *Delete a wall* (on page 265)

Place a wall by 2-D sketch

1. Click **Place Wall** on the vertical toolbar.
2. Define the sketching plane. The sketching plane defines the bottom elevation of the wall.

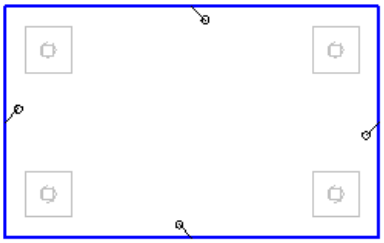


3. Click **Sketch 2D**.
4. Select objects in the model that you want to see when sketching the wall path in the 2-D environment.



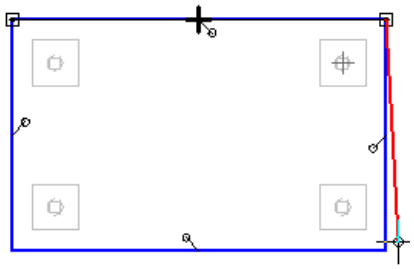
5. Click **Draw**.

TIP The 2D sketch orientation is based on the active coordinate system as defined in the **Pin Point** command.

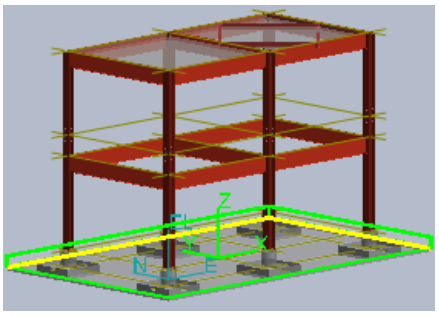


6. In the 2-D environment, draw the path of the wall.

TIP You can draw more than one wall path at a time in the 2-D environment.

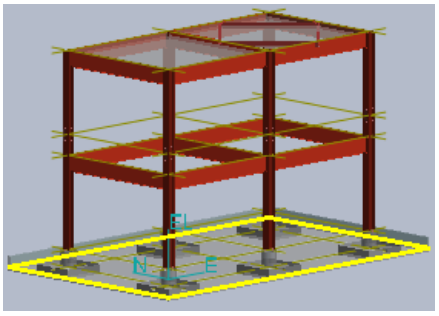


7. In the 2-D environment, click **Close**.
8. Click **Finish**.



9. Set the wall system, type, composition, and other properties.
10. Optionally, click **Select Boundaries** to limit the height of the wall.
11. Click **Finish**.

    The software places the walls in the model.



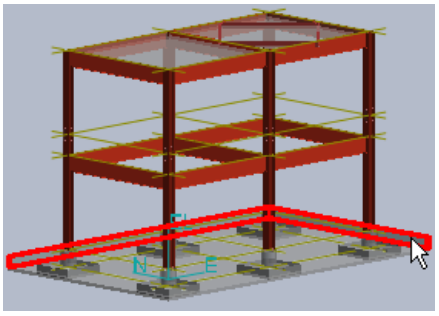
12. The software maintains the same sketching plane and prompts you to select references for the next wall. Go back to step 4.

    -OR-

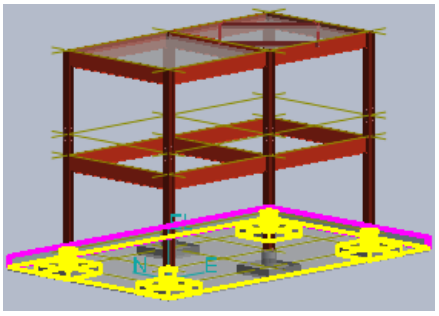
    Click **Cancel** to return to the main ribbon to define a new sketching plane. Go back to step 2.

Add end segment to existing wall using 2-D sketch

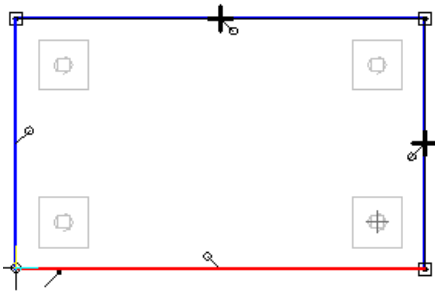
1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Wall Systems**.
3. Select the wall to which to add another wall segment.



4. Click **Select a Path** on the ribbon.
5. Click **Sketch 2D** on the ribbon.
6. Select objects in the model to see when sketching the wall path in the 2-D environment.

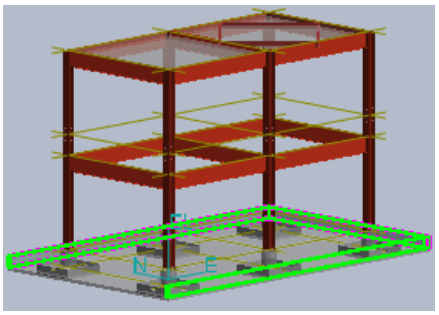


7. Click **Draw**.
8. In the 2-D environment, add on to the path of the existing wall path.



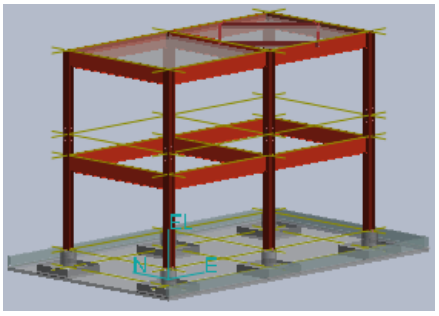
9. In the 2-D environment, click **Close**.

10. Click **Finish**.



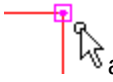
11. Set the wall system, type, composition, and other properties.
12. Optionally, click **Select Boundaries** to limit the height of the wall.
13. Click **Finish**.

    The software places the new wall segments in the model.



Place split walls by 2-D sketch

1. Click **Place Wall** on the vertical toolbar.
2. Define the sketching plane. The sketching plane defines the bottom elevation of the wall.
3. Click **Sketch 2D**.
4. Select objects in the model that you want to see when sketching the wall paths in the 2-D environment.
5. Click **Draw**.
6. In the 2-D environment, turn off (no check mark) **Tools > Maintain Relationships**. By clearing this option, each line segment that you draw becomes its own wall.

   TIP As an alternative to turning off Maintain Relationships, you can delete the relationship handles  at the segment end points after you have finished placing the segments in order to create multiple walls.
7. In the 2-D environment, draw the path of the walls.
8. In the 2-D environment, click **Close**.
9. Click **Finish**.

10. Set the wall system, type, composition, and other properties.
11. Optionally, click **Select Boundaries** to limit the height of the wall.
12. Click **Finish**.

    The software places the walls in the model.
13. The software maintains the same sketching plane and prompts you to select references for the next wall. Go back to step 4.

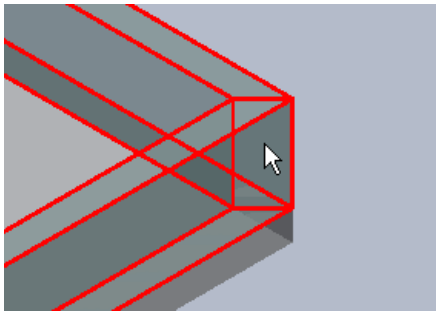
    -OR-

    Click **Cancel** to return to the main ribbon to define a new sketching plane. Go back to step 2.

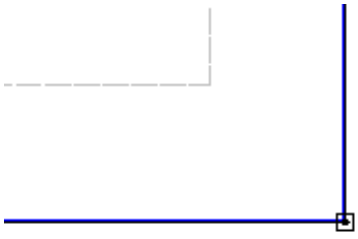
NOTE If you draw more than one wall path in the 2-D environment, the walls will move as a single unit if you move any of the walls regardless of whether or not the walls are connected to each other. If you will need to move any of the walls individually, you must draw that wall's path in the 2-D environment by itself.

Add fillet corner to existing wall using 2-D sketch

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Wall Systems**.
3. Select the wall to which to add the fillet corner.

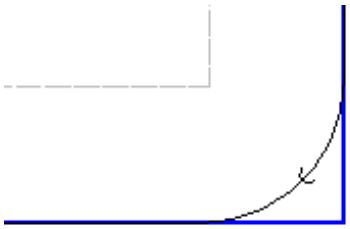


4. Click **Select a Path** on the ribbon.
5. Click **Sketch 2D** on the ribbon.
6. Click **Draw**.

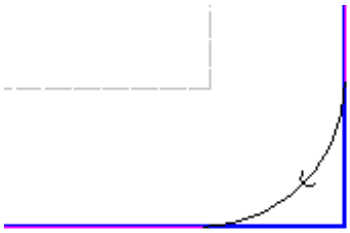


7. Click **Fillet** on the vertical toolbar.

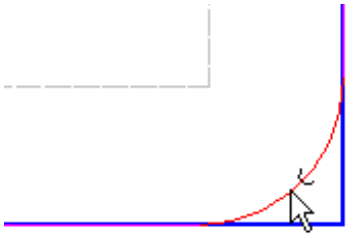
8. Place the fillet on the wall path by identifying the two sides for the fillet and then the radius.



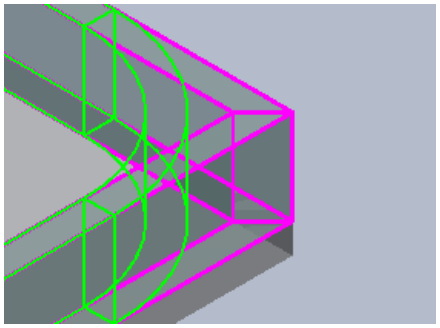
9. Click **Add or remove graphic from a group** on the **Change** toolbar.
10. Click **Select Graphic Group** on the ribbon.
11. Select the original wall path. Be sure to use the **Bottom Up** option during selection.



12. Click **Select elements to add to group or remove from it** on the ribbon.
13. Select the fillet segment of the wall path that you just placed. Be sure to use the **Bottom Up** option during selection.



14. Click **Add** on the ribbon.
15. In the 2-D environment, click **Close**.
16. Click **Finish**.



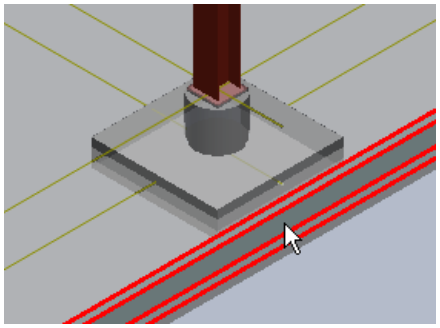
17. Set the wall system, type, composition, and other properties.
18. Optionally, click **Select Boundaries** to limit the height of the wall.
19. Click **Finish**.

    The software places the fillet in the wall.



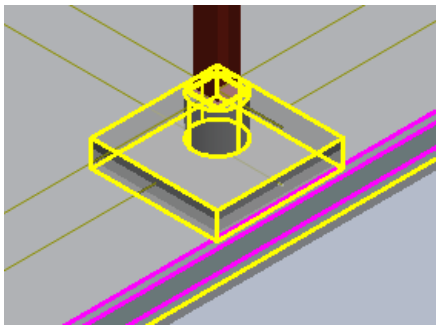
Add middle segments to existing wall using 2-D sketch

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Wall Systems**.
3. Select the wall to which to add segments.

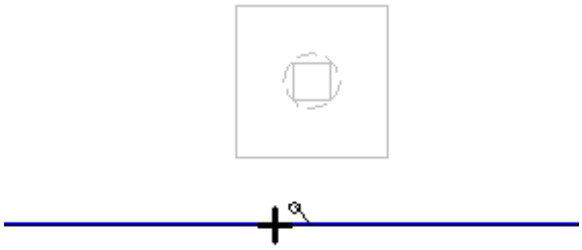


4. Click **Select a Path** on the ribbon.
5. Click **Sketch 2D** on the ribbon.

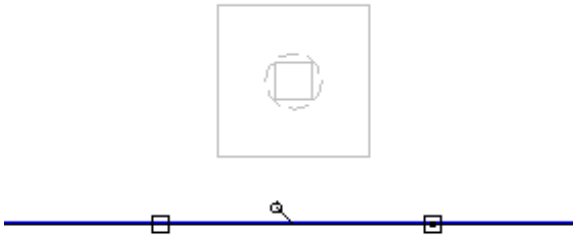
6. Select objects in the model that you want to see when sketching the wall path in the 2-D environment.



7. Click **Draw** .



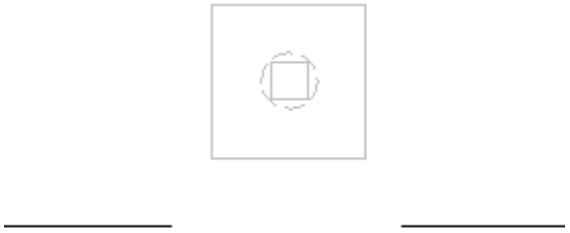
8. Click **Split** on the vertical toolbar.
9. Identify the two locations on the wall between which you want to add to the wall. Be sure to use the **Bottom Up** option during selection.



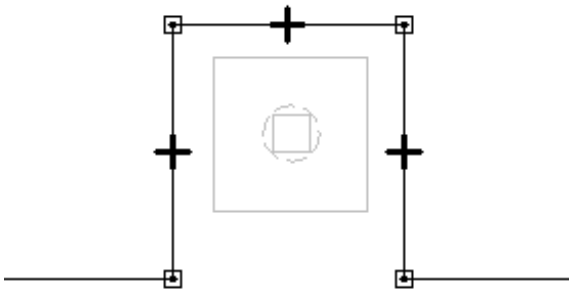
10. Click **Add or remove graphic from a group** on the **Change** toolbar.
11. Click **Select Graphic Group** on the ribbon.
12. Select the original wall path. Be sure to use the **Bottom Up** option during selection.
13. Click **Select elements to add to group or remove from it** on the ribbon.
14. Select the segment of the wall path between the two points that you identified during the **Split** command. Be sure to use the **Bottom Up** option during selection.
15. Click **Remove** on the ribbon. You have now removed the segment from the wall path group.

16. Click **Select** on the vertical toolbar.
17. Select the segment between the two points that you just removed from the group. Be sure to use the **Bottom Up** option during selection.
18. Press **Delete** on the keyboard.

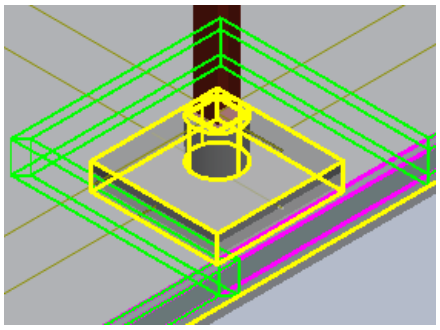
    You should now have a wall path with a gap in it.



19. Using commands on the vertical toolbar, like the **Line** command, draw the new wall segment in the gap. Make sure you connect to the two ends of the gap.

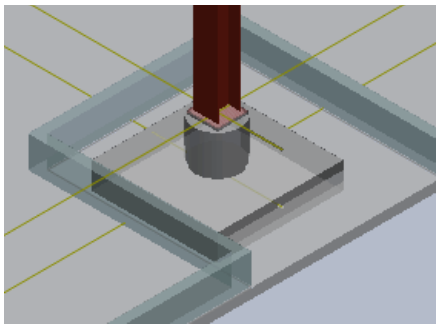


20. In the 2-D environment, click **Close**.
21. Click **Finish**.



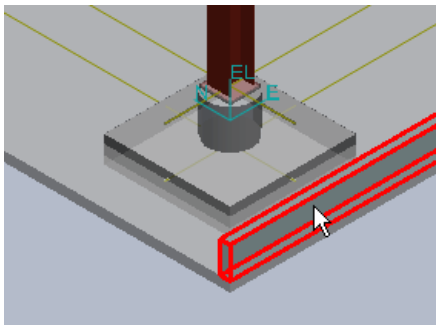
22. Set the wall system, type, composition, and other properties.
23. Optionally, click **Select Boundaries** to limit the height of the wall.
24. Click **Finish**.

The software places the new wall segments in the model.

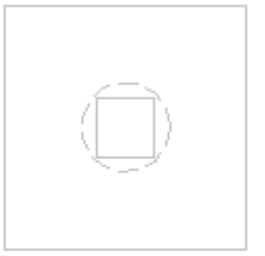


Modify wall length using 2-D sketch

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Wall Systems**.
3. Select the wall to modify.

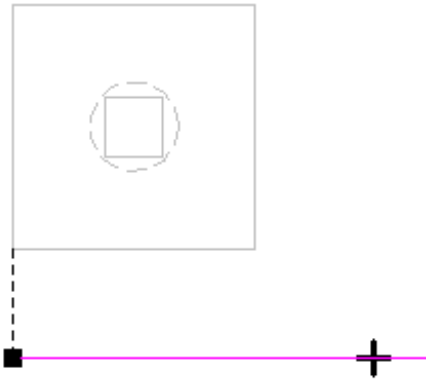


4. Click **Select a Path** on the ribbon.
5. Click **Sketch 2D** on the ribbon.
6. Select objects in the model that you want to see when modifying the wall length in the 2- D environment.
7. Click **Draw**.

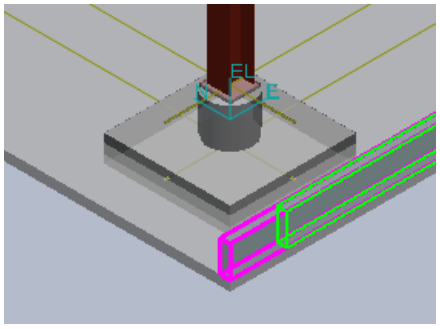




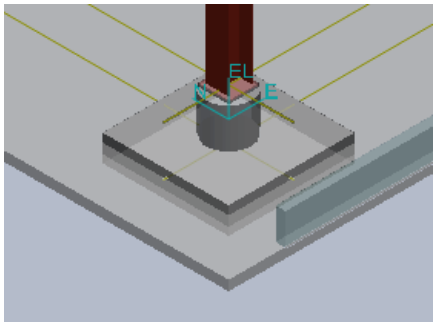
8. Click **Bottom Up** on the ribbon.
9. Select the wall segment to modify.
10. In the **Length** box, key in the new length for the segment, or grab the box at the end of the segment and move it to the new location.



11. In the 2-D environment, click **Close**.
12. Click **Finish**.



13. Optionally, edit the wall system, type, composition, and other properties.
14. Optionally, click **Select Boundaries** to limit the height of the wall.
15. Click **Finish**.



Create gap in existing wall using 2-D sketch

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Wall Systems**.
3. Select the wall that you want to place the gap in.
4. Click **Select a Path** on the ribbon.
5. Click **Sketch 2D** on the ribbon.
6. Select objects in the model that you want to see in the 2-D environment.
7. Click **Draw**.
8. Click **Split** on the vertical toolbar.
9. Identify the two locations on the wall between which you want to place the gap. Be sure to use the **Bottom Up** option during selection.
10. Click **Add or remove graphic from a group** on the **Change** toolbar.
11. Click **Select Graphic Group** on the ribbon.
12. Select the original wall path. Be sure to use the **Bottom Up** option during selection.
13. Click **Select elements to add to group or remove from it** on the ribbon.
14. Select the segment of the wall path where the gap should be.
15. Click **Remove** on the ribbon.
16. Click **Select** on the vertical toolbar.
17. Select the segment between the two points that you just removed from the group. Be sure to use the **Bottom Up** option during selection.
18. Press **Delete** on the keyboard.

    You should now have a wall path with a gap in it.
19. Click **Add or remove graphic from a group** on the **Change** toolbar.
20. Click **Select Graphic Group** on the ribbon.
21. Select the original wall path.
22. Click **Select elements to add to group or remove from it** on the ribbon.
23. Select all the segments on one side of the gap. Be sure to use the **Bottom Up** option during selection.
24. Click **Remove** on the ribbon.

    By removing all the segments on one side of the gap from the original wall group, you create a new second wall. The software currently does not support having a gap in a single wall.
25. In the 2-D environment, click **Close**.
26. Click **Finish**.

27. Set the wall system, type, composition, and other properties.
28. Optionally, click **Select Boundaries** to limit the height of the wall.
29. Click **Finish**.

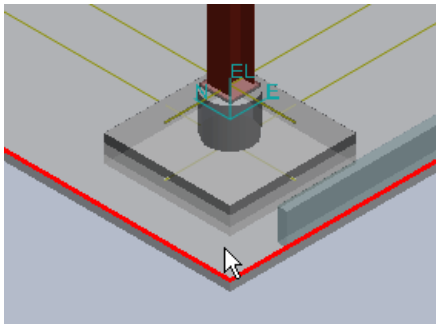
    The software removes the segment from the wall thus creating the gap.

Rotate an existing wall using 2-D sketch

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Wall Systems**.
3. Select the wall that you want to rotate.
4. Click **Select a Path** on the ribbon.
5. Click **Sketch 2D** on the ribbon.
6. Select objects in the model that you want to see when sketching the wall path in the 2-D environment.
7. Click **Draw**.
8. Click **Select** on the vertical toolbar. Be sure to use the **Bottom Up** option during selection.
9. Select the wall path that you want to rotate.
10. In the **Change** toolbar, click **Rotate**.
11. Rotate and move the wall as needed.
12. In the 2-D environment, click **Close**.
13. Click **Finish**.
14. Set the wall system, type, composition, and other properties.
15. Optionally, click **Select Boundaries** to limit the height of the wall.
16. Click **Finish**.

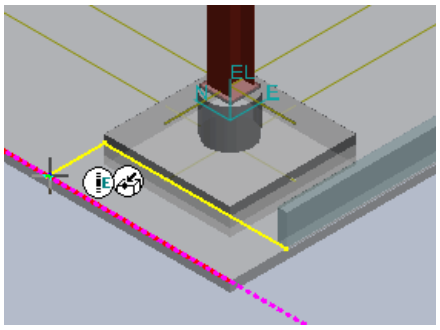
Place a wall by 3-D sketch

1. Click **Place Wall** on the vertical toolbar.
2. Define the sketching plane.

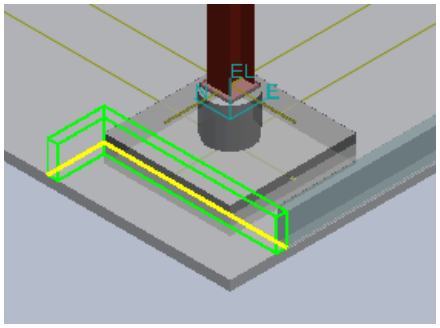


3. Click **Sketch 3D**.
4. Sketch a path that defines the position of the wall.

   Define a Path (on page 501)

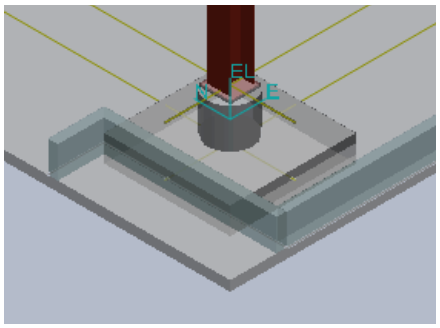


5. Click **Finish**.



6. Set the wall system, type, composition, and other properties.
7. Optionally, click **Select Boundaries** to limit the height of the wall.
8. Click **Finish**.

The software places the wall in the model.



9. The software maintains the same sketching plane and resets itself for you to define the next wall path. Go back to step 4.

   -OR-

   Click **Cancel** to return to the main ribbon to define a new sketching plane. Go back to step 2.

Place split walls by 3-D sketch

1. Click **Place Wall** on the vertical toolbar.
2. Define the sketching plane. The sketching plane sets the bottom elevation of the wall.
3. Click **Sketch 3D** 3D.
4. In the 3-D environment, draw the path of the wall.
5. When you are ready to split the wall path to create a new, adjoining wall, select **Split** as the **Turn Type**.

   NOTES

   - The **Split** turn type option is available after you place a second vertex point on the wall path.
   - You cannot select the middle point of an arc to insert a split.
6. You can insert another split to add another new wall system. After the **Split** option is selected, it remains set as the **Turn Type** option.
7. Click **Finish** to complete the 3-D sketch.
8. Set wall system, type, composition, and other properties for each wall, or use the default properties.
9. Click **Finish**.

   The software places the walls in the model. The separate wall systems you created are listed in the Workspace Explorer.

Split an existing wall using 3-D sketch

1. Click **Select** on the vertical toolbar.
2. Select the wall you want to split.

   NOTE The wall system must contain at least one vertex to insert a split.
3. Click **Select a Path** on the ribbon.
4. Click **Sketch 3D** on the ribbon.
5. Click on the wall vertex where you want to insert the split.

   NOTE You cannot insert a split on the middle point of an arc.
6. Select **Split** as the **Turn Type** on the ribbon.
7. Click **Finish** to complete the 3-D sketch.
8. The properties of the initial wall are used for the walls you create with the **Split** option. You can modify the wall properties for each wall.
9. Click **Finish**.

   The software creates two separate wall systems that can be edited independently.

Add end segment to existing wall using 3-D sketch

1. Click **Select** on the vertical toolbar.
2. Set the **Locate Filter** to **Wall Systems**.
3. Select the wall that you want to add to.
4. Click **Select a Path** on the ribbon.
5. Click **Sketch 3D**.
6. Click **Create** on the ribbon.
7. Select one of the boxes on the existing wall path as your starting point. If you can add to the wall path from that path point, the box will turn red to indicate that you have selected it. If the box does not turn red, you cannot add on to the wall from that path point.
8. Define the addition to the path using the path commands. For more information, see *Define a Path* (on page 501).
9. Click **Finish**.
10. Edit the wall system, type, composition, and other properties if needed.
11. Optionally, click **Select Boundaries** to limit the height of the wall.
12. Click **Finish**.

    The software places the wall extension in the model.

Modify wall cross-section

1. Click **Select** on the vertical toolbar.
2. Select **Wall Systems** in the locate filter.
3. Select the wall to edit.
4. On the ribbon bar, select a new cross-section using the **Composition** option.

Edit wall properties

1. Click **Select** on the vertical toolbar.
2. Select **Wall Systems** in the locate filter.
3. Select the wall to edit.
4. Click **Edit > Properties**.
5. Edit the wall properties as needed. For more information about the properties, see *Wall System Properties Dialog Box* (on page 265).

Edit multiple walls

1. Click **Select** on the vertical toolbar.
2. Select **Wall Systems** in the locate filter.
3. Select multiple wall systems in a graphics window or from the Workspace Explorer.
4. Click **Edit > Properties**.
5. Edit the wall properties as needed. The edits you make *affect all selected walls*.

NOTES

- You can select multiple wall parts and edit them as a group. You can edit properties on the ribbon, or click **Properties** to open the **Wall Properties** dialog box and change values.
- Property fields that have different values are left blank in the ribbon and in the properties dialog box. Only property values that are same for the selected walls display.
- The **Sketching Plane**, **Select a Path** and **Select Boundaries** options in the ribbon are unavailable and cannot be edited when multiple walls are selected.
- When you edit a property for multiple wall parts, *the new value is set for ALL the selected walls*.

Delete a wall

1. Click **Select** on the vertical toolbar.
2. Select **Wall Systems** in the **Locate Filter**.
3. Select the wall to delete.
4. Click **Delete**.

Wall System Properties Dialog Box

Specifies the properties for the wall system that you are editing. You can define the default values in the **StructWallGeneral.xls** and **StructWallLayer.xls** workbooks.

See Also

General Tab (Wall System Properties Dialog Box) (on page 265)
Layer Tab (Wall System Properties Dialog Box) (on page 267)
Section Tab (Wall System Properties Dialog Box) (on page 267)
Edit Wall Properties (on page 264)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

General Tab (Wall System Properties Dialog Box)

The **General** tab displays the wall properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one wall and then selected the properties command, only the common properties between the selected walls display.

When viewing properties for a single wall, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on properties. You can access this document using the **Help > Printable Guides** command in the software.

Category

Select the properties that you want to view for the wall. Wall properties are divided into several different categories: **Standard**. You select the category to define values for by using the **Category** option.

Standard

The standard properties that display depend on the reference data for the wall. Because wall reference data is fully customizable, only the common properties are documented.

Parent System

Select the system to which the wall belongs.

Name

Displays the name of the wall. The wall name is based on the **Naming Rule** selection. If you want to type a new name for the wall, in the **Naming Rule** box, select **User Defined**, and

then type a name for the wall in the **Name** box.

Naming Rule

Specify the naming rule that you want to use to name this wall.

Default Name Rule - Names the wall using the format "WallSystem-<location>-<index>" where <location> is the global workshare location ID and <index> is a unique index number that starts at 0001. For example, WallSystem-1-0043.

Unique Name Rule - Names the wall using the format "<system>-WallSystem-<location>-<index>" where <system> is the name of the parent system to which the wall system belongs, <location> is the global workshare location ID and <index> is a unique index number that starts at 0001. For example, Structure System-WallSystem-1-0043.

User Defined - Select to specify the wall system name yourself in the **Name** box.

Type

Select the wall type.

Composition

Select the composition that you want to use for the wall type. The layers in the select composition appear.

Priority

Specifies the priority assigned to the object system. Smart 3D uses the priority to group objects. To change the options on the list, edit the **Structural Member Priority** select list in Catalog.

Continuity Type

Indicates the way in which the object system reacts when the system intersects another object system (your automatic splitting preference). The list is defined by the StructContinuity codelist.

Continuous indicates that the object system should split the other object system.

Intercostal indicates that the object system should be split by the other object system.

You cannot split objects that have a **Continuity Type** setting of **Continuous**.

Continuity Priority Number

Specifies the priority when two profile systems that have the same **Continuity** setting intersect. The profile system with the lowest continuity priority number penetrates the other system. For example, two profile systems intersect and the **Continuity** setting of each set to **Continuous**. The profile system with the lowest **Continuity Priority** number splits the other profile system. You can only modify this property at the root system.

Bearing

Indicates whether or not the wall is a load bearing wall.

Boundary Offset

Enter the distance the wall should be offset from the selected boundary.

Layer Tab (Wall System Properties Dialog Box)

Displays the layers for the selected wall type on the **General** tab. Layers are defined in the **StructWallLayer.xls** workbook. Refer to the *Structure Reference Data Guide for* more information about this workbook. You can access this document using the **Help > Printable Guides** command in the software.

Composition

Select the composition that you want to use for the wall type. The layers in the select composition appear.

Total Thickness

Displays the total thickness of all the layers in the wall.

Layer

Select the layer of the composition to view or edit properties for.

Property

Displays the defined properties for the active layer.

Value

Specify a value for the layer property.

See Also

Wall System Properties Dialog Box (on page 265)

Section Tab (Wall System Properties Dialog Box)

The **Section** tab displays the properties for the wall's cross-section. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one wall and then selected the properties command, only the common properties between the selected walls display.

When viewing properties for a single wall, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on properties. You can access this document using the **Help > Printable Guides** command in the software.

Category

Select the properties that you want to view for the cross-section. Cross-section properties are divided into several different categories: **Standard**. You select the category to define values for by using the **Category** option.

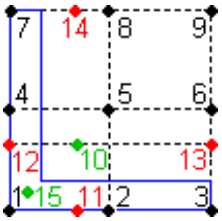
Standard

The standard properties that display depend on the reference data for the wall. Because wall reference data is fully customizable, only the common properties are documented.

Wall Cross Sections Name

Displays the name of the wall cross-section.

Position



Specify the relative position of the wall to the path. Fifteen positions are available. The location of positions 10 (center-of-gravity) and 15 (shear center) depend on the wall's section shape. The local z- axis of the wall and the center-of-gravity point of the section define positions 11 and 14. The local y-axis of the wall and the center-of-gravity point of the section define positions 12 and 13.

Horizontal Offset

Specifies the horizontal offset between the object and the object's selected **Position** location.

Vertical Offset

Specifies the vertical offset between the object and the object's selected **Position** location.

Thickness

Type the thickness of the wall.

Height

Type the height of the wall.

Reflect

Reflects or mirrors the cross-section about the wall's path. This parameter affects both symmetric and asymmetric sections.

Angle

Enter the angle of the wall from vertical. You can enter both positive and negative angles.

CentroidX

Type the horizontal distance from the designated member edge to the centroidal axis.

CentroidY

Type the vertical distance from the designated member edge to the centroidal axis.

Moment of Inertia about X (Ixx)

Type the moment of inertia for the section's local x-axis.

Moment of Inertia about Y (Iyy)

Type the moment of inertia for the section's local y-axis.

Plastic Section Modulus for X (Zxx)

Type the plastic section modulus for the section's local x-axis.

Elastic Section Modulus about X (Sxx)

Type the section modulus for the section's local x-axis.

Radius of Gyration about the X-axis (Rxx)

Type the radius of gyration for the section's local x-axis.

Plastic Section Modulus about Y (Zyy)

Type the plastic section modulus for the section's local y-axis.

Elastic Section Modulus about Y (Syy)

Type the section modulus for the section's local y-axis.

Radius of Gyration about the Y-axis (Ryy)

Type the radius of gyration for the section's local y-axis.

Torsional Moment of Inertia (J)

Type the torsional moment of inertia for the section.

Warping Constant (Cw)

Type the warping constant for the section.

Polar Radius of Gyration about the Shear Center (ro)

Type the polar radius of gyration about the shear center.

Flexural Constant (H)

Type the flexural constant for the section.

Wall Part Properties Dialog Box

Specifies the properties for the wall part that you are editing. You can define the default values in the **StructWallGeneral.xls** and **StructWallLayer.xls** workbooks.

Editing Multiple Wall Parts

You can select multiple wall parts and edit them as a group. When editing, keep the following in mind:

- When you edit a property for multiple wall parts, the new value is set for ALL the selected walls.
- Property fields that have different values are left blank in all the Property tabs. Only properties where the values are the same for the selected walls display.

NOTE While you can edit parts, you cannot delete parts.

See Also

General Tab (Wall Part Properties Dialog Box) (on page 269)
Layer Tab (Wall Part Properties Dialog Box) (on page 273)
Section Tab (Wall Part Properties Dialog Box) (on page 273)
Edit Wall Properties (on page 264)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

General Tab (Wall Part Properties Dialog Box)

The **General** tab displays the wall properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one wall and then selected the properties command, only the common properties between the selected walls display.

When viewing properties for a single wall, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on properties. You can access this document using the **Help > Printable Guides** command in the software.

Category

Select the properties that you want to view for the wall. Wall properties are divided into several different categories: **Standard**, **Weight and CG**, and **Dimensions**. You select the category to define values for by using the **Category** option.

Standard

The standard properties that display depend on the reference data for the wall. Because wall reference data is fully customizable, only the common properties are documented.

Parent System

Select the system to which the wall belongs.

Name

Displays the name of the wall. The wall name is based on the **Naming Rule** selection. If you want to type a new name for the wall, in the **Naming Rule** box, select **User Defined**, and then type a name for the wall in the **Name** box.

Naming Rule

Specify the naming rule that you want to use to name this wall.

- **Default Name Rule** - Names the wall using the format "WallPart-<location>-<index>" where <location> is the global workshare location ID and <index> is a unique index number that starts at 0001. For example, WallPart-1-0043.
- **Unique Name Rule** - Names the wall using the format "WallPart-<location>-<index>" where <location> is the global workshare location ID and <index> is a unique index number that starts at 0001. For example, WallPart-1-0043.
- **User Defined** - Select to specify the wall part name yourself in the **Name** box.

Composition

Displays the composition of the wall.

Reporting Requirement

Specify whether or not this wall part is reported.

Reporting Type

Select the reporting requirements code for the wall part. Valid selections are defined in the **Catalog** task in the **Reporting Type** select list.

Dimensions

Total Volume

Specifies the volume of the slab or wall (length × width × height).

Net Volume

Specifies the volume of the wall minus the volumes of all openings in the wall.

Total Surface

Specifies the surface area of the slab or wall (top surface plus the bottom surface plus the side surfaces × length).

Net Surface

Specifies the surface area of the slab or wall minus the surface area of all the openings in the slab or wall.

Projected Surface

Displays the surface area of the wall's bottom surface.

Net Projected Surface

Displays the surface area of the wall's bottom surface minus the surface area of all openings in the wall.

Length

Displays the wall length.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material density multiplied by the object's solid volume. Therefore, the material of the object affects the weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

NOTE For equipment, the **Weight and CG** property **Wet Weight** is the sum of **Dry Weight** and **Water Weight**. The dry weight and water weight values are catalog properties entered on the part sheet for the equipment.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Wet CG X

Specifies the X-axis location of the wet center-of-gravity.

Wet CG Y

Specifies the Y-axis location of the wet center-of-gravity.

Wet CG Z

Specifies the Z-axis location of the wet center-of-gravity.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

Layer Tab (Wall Part Properties Dialog Box)

Displays the layers for the selected wall type on the **General** tab. Layers are defined in the **StructWallLayer.xls** workbook. Refer to the *Structure Reference Data Guide for* more information about this workbook. You can access this document using the **Help > Printable Guides** command in the software.

Composition

Select the composition that you want to use for the wall type. The layers in the select composition appear.

Total Thickness

Displays the total thickness of all the layers in the wall.

Layer

Select the layer of the composition to view or edit properties for.

Property

Displays the defined properties for the active layer.

Value

Specify a value for the layer property.

See Also

Wall Part Properties Dialog Box (on page 269)

Section Tab (Wall Part Properties Dialog Box)

The **Section** tab displays the properties for the wall's cross-section. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one wall and then selected the properties command, only the common properties between the selected walls display.

When viewing properties for a single wall, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on properties. You can access this document using the **Help > Printable Guides** command in the software.

Category

Select the properties that you want to view for the cross-section. Cross-section properties are divided into several different categories: **Standard**. You select the category to define values for by using the **Category** option.

Standard

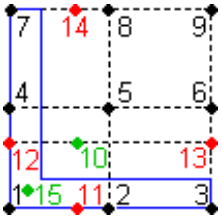
The standard properties that display depend on the reference data for the wall. Because wall reference data is fully customizable, only the common properties are documented.

Wall Cross Sections Name

Displays the name of the wall cross-section.

Position

Specify the relative position of the wall to the path. Fifteen positions are available. The location of positions 10 (center-of-gravity) and 15 (shear center) depend on the wall's section shape. The local z- axis of the wall and the center-of-gravity point of the section define positions 11 and 14. The local y-axis of the wall and the center-of-gravity point of the section define positions 12 and 13.



Horizontal Offset

Specifies the horizontal offset between the object and the object's selected **Position** location.

Vertical Offset

Specifies the vertical offset between the object and the object's selected **Position** location.

Thickness

Defines the insulation thickness.

Height

Type the height of the wall.

Reflect

Reflects or mirrors the cross-section about the wall's path. This parameter affects both symmetric and asymmetric sections.

Angle

Enter the angle of the wall from vertical. You can enter both positive and negative angles.

Wall Connection Properties Dialog Box

Specifies the properties for the wall assembly connection that you are editing. You can define the default values in the **StructWallAssemblyConnections.xls** workbook.

See Also

Occurrence Tab (Wall Connection Properties Dialog Box) (on page 274)
Definition Tab (Wall Connection Properties Dialog Box) (on page 276)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

Occurrence Tab (Wall Connection Properties Dialog Box)

The **Occurrence** tab displays the assembly connection properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one assembly connection, and then selected the properties command, only the common properties between the selected assembly connections display.

When viewing properties for a single assembly connection, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on properties. You can access this document using the **Help > Printable Guides** command in the software.

Name

Displays the name of the assembly connection. The assembly connection name is based on the **Name Rule** selection. If you want to type a new name for the assembly connection, in the **Name Rule** box, select **User Defined**, and then type a name for the assembly connection in the **Name** box.

Name Rule

Specify the naming rule that you want to use to name this assembly connection.

- **Default Name Rule** - Names the assembly connection using this format: <assembly connection>-<location>-<unique index> where <assembly connection> is the name of the assembly connection defined in the catalog, <location> is the global workshare location, and <unique index> is an index number that starts at 0001. For example, WPTrimByPointAtEnd3-1-0045.
- **Unique Name Rule** - Names the assembly connection using this format: <parent system name>-<assembly connection>-<location>-<unique index> where <parent system name> is the parent System selected for the assembly connection, <assembly connection> is the name of the assembly connection defined in the catalog, <location> is the global workshare location, and <unique index> is an index number that starts at 0001. For example, Structure System-WPTrimByPointAtEnd3-1-0045.
- **User Defined** - Select to specify the assembly connection name yourself in the **Name** box.

System

Displays the name of the Wall Part to which the assembly connection belongs.

Boundary Size

Select whether the boundary is infinite or finite.

Boundary Type

Select the boundary type: above, below, start, end, or contact.

Connection Type

Select the connection type: aligned, along, or corner.

Offset

Enter the offset distance between the wall and the boundary.

Reference

When a member is used as a boundary for the wall, you can use this option to specify which part of the member the wall should use as the boundary: near, far, or center.

- Select near when you want the wall to stop at the inner most plane of the bounding member.
- Select center when you want the wall to stop at the member-line axis of the bounding member.

- Select far when you want the slab to stop at the out-most plane of the bounding member.

Definition Tab (Wall Connection Properties Dialog Box)

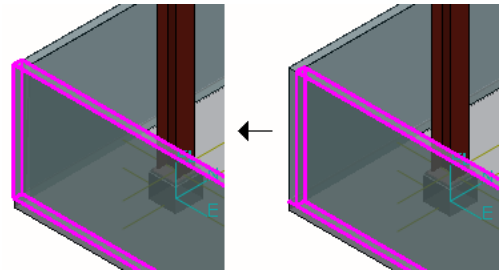
The **Definition** tab displays the wall assembly connection properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one wall assembly connection, and then selected the properties command, only the common properties between the selected wall assembly connections display.

The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties. You can access this document using the **Help > Printable Guides** command in the software.

SECTION 16

Toggle Wall Corner

When two separate walls join at a corner, one wall extends past the other wall to form the corner. You can use this command to toggle which wall extends past the other.



1. Click **Toggle Wall Corner** .
2. Select the first wall at the corner.
3. Select the other wall at the corner.
4. Click **Finish**.

SECTION 17

Place Equipment

 Specifies any piece of equipment from the Equipment folder of the catalog and places an occurrence of it inside the model. You can modify the offset of the equipment, its relationships to other equipment, and other properties during or after placement. Using positioning relationships, the **Place Equipment** command allows you to mate, connect, or align equipment, and you can use common tools like the **PinPoint** command for precise positioning of the equipment.

If a parent object is selected when you start **Place Equipment**, the **System** property is automatically updated with the parent object.

You place equipment objects into the model by selecting the equipment from the **Select Equipment** dialog box and positioning the equipment in the model using the available relationships. When you select an equipment object from the Catalog, you can define a default surface so that, when the equipment is placed into the model, the software automatically creates a relationship to any other surface or reference element that you select. For example, you might define the bottom of a pump as the default surface because you want the software to mate that surface to the floor. If the software cannot find a suitable mating surface for the equipment, the equipment object is placed in free space, pending additional design of the structures or other reference elements.

You can also provide an optional offset distance from the surface or reference elements. The default offset distance for any new piece of equipment is zero or the last offset used in the current session. The software maintains the offset relationship between the default surface and the surface or reference element in the workspace in the event of any changes to their position (for example, if a slab is lowered, then the equipment follows the slab, maintaining the offset). You can further define the equipment position and orientation by mating, aligning, or connecting equipment to other objects in the model, or by moving or rotating the equipment.

In addition to placing equipment from the **Select Equipment** dialog box, you can also drag and drop equipment directly from the **Catalog Browser** in another session opened on the same computer. In this case, the software places the equipment in free space, with relationship to other elements in the model made by locating objects while dragging the equipment in the new session.

Cable and Equipment

If you are going to use an object for routing cable, you must not model that object as equipment.

Although it may sometimes be desirable to model a piece of raceway, cable tray, or cable bus as a piece of equipment, you should not do this if there is any possibility of ever needing to route cable through the object. This is because cable routes cannot traverse equipment; they can only be terminated by equipment. A cable cannot be properly routed through a piece of equipment. The cable can only interconnect two pieces of equipment.

An example is a pull box. This object should be created in the catalog data as a conduit fitting, not as a piece of equipment. Otherwise, the object will render conduit networks useless for cable routing every time it is used.

Equipment with Occurrence Properties Modeled Using Solid Edge

You cannot place equipment with occurrence properties modeled using Solid Edge unless Solid Edge is installed on your computer. If you try to place such an object, the following message appears: **Cannot start Solid Edge. Exiting InitializSymbolDefinition.**

As a workaround to the Solid Edge requirement, you can use the Bulkload utility with the flavors option. Designers can create dozens of variations for any equipment part imaginable. Also, by creating several variations of a part, rather than using occurrence properties to create the variations automatically, designers can implement custom Solid Edge equipment without having to install Solid Edge on every designer's computer.

Each variation of a part represents some minor deviation from the catalog part, be it on the basis of size, operational specifications, or material. With Solid Edge, new variations can be created on the basis of size, while the software allows manipulation of the operating parameters or material of construction in the reference data. For more information on bulkloading with flavors, refer to the *Reference Data Guide*, accessible from the **Help** > **Printable Guides** command in the software.

Place Equipment Ribbon

Sets options for adding equipment to your model. This ribbon appears automatically after you select the **Place Equipment** command and then select an equipment object, or when you select an existing piece of equipment.

TIP To find out the name of an option on the ribbon, pause the pointer over an option and read the ToolTip.

Equipment Properties

Edits the occurrence properties and reviews the static properties of an existing piece of equipment. Equipment properties can be set only after an equipment object is placed in the model. Equipment properties can be edited only after the object is placed in the model.

NOTE Any object modeled in Solid Edge that has occurrence properties cannot be placed or modified unless your computer has a copy of Solid Edge installed.

User Defined Form Definition

Displays **User Defined Form** for the particular equipment if it is defined in the catalog. See the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

Relationship List

Lists all relationships for the selected equipment and provides an option for creating a new relationship if the equipment is not already fully constrained. An equipment part is fully constrained when it has sufficient defined relationships to prevent movement or rotation of the part along all three coordinate axes.

Positioning Relationships

Displays the available options for types of positioning relationships. Some options may not be available for all equipment types. See Positioning Relationships for more information.

Delete Relationship

Removes the selected relationship from the equipment model and the database. Using the **Relationships** list box, select a previously existing relationship for the equipment, and click

Delete Relationship. You can use this command only when modifying existing equipment.

Equipment Reference

Prompts you for the reference on the equipment to be placed that will be affected by the positioning relationship. In all cases, the part that you select in this step moves to create the relationship, and the part chosen in the **Second Part Reference** step remains fixed.

Second Part Reference

Prompts you for the reference on the equipment object or reference element already in the model that will be affected by the positioning relationship. After you select the reference, the software repositions the first equipment part chosen with respect to the second part selected in the definition of the relationship.

Offset

Defines the offset distance for a mate or align relationship. Offsets are disabled when establishing a connect relationship. You can adjust this value after initial equipment placement if needed.

Name

Displays the equipment name, as dictated by your predefined name rules, and accepts changes to that name.

System

Specifies the system with which to associate the selected equipment. The default system is the Model itself.

<Select Graphically>

Allows you to select the system from the graphic view or from the **Workspace Explorer**. You can then place or modify the equipment.

Select Equipment Dialog Box

Specifies the equipment needed for placement. This dialog box appears automatically when you click the **Place Equipment** command. By browsing through the part hierarchy, you can find any piece of equipment in the Catalog database. After you select a piece of equipment and click **OK**, the **Equipment Properties** dialog box appears so that you can define properties for the new equipment. After you click **OK** on the **Properties** dialog box, the software returns you to the model so that you can finalize configuration and placement.

Save

This option is unavailable.

Cut

This option is unavailable.

Copy

This option is unavailable.

Paste

This option is unavailable.

Delete

This option is unavailable.

Refresh

This option is unavailable.

Insert Row

This option is unavailable.

Move Up

This option is unavailable.

Move Down

This option is unavailable.

Properties

Displays the equipment properties as defined in the catalog.

Preview

Displays a bitmap symbol of the selected equipment. The image file must be assigned to the equipment in the catalog reference data.

Filter

This option is unavailable.

Sort

This option is unavailable.

Customize Current View

This option is unavailable.

List View

Sets the dialog box to display equipment in a list view.

Grid View

Sets the dialog box to display equipment in a spreadsheet-style grid view.

Back

Returns you to the previously selected equipment part or node. Use this command to navigate through the equipment hierarchy to the specific part you need.

Forward

Sends you to the last selected equipment part or node that you moved away from by using the **Back** button. Use this command to navigate through the equipment hierarchy to the specific part you need.

Up One Level

Brings up the next highest level of the Equipment catalog hierarchy. Use this command to navigate through the equipment hierarchy to the specific part you need.

Check Data

This option is unavailable.

Check Data Ignored Inconsistencies

This option is unavailable.

Help

Displays on-line help for Equipment and Furnishings.

Address

Specifies your exact location within the displayed hierarchy.

What do you want to do?

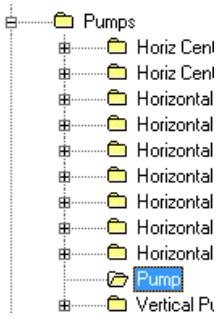
- *Place equipment from the catalog* (on page 282)
- *Set positioning relationships for equipment* (on page 283)
- *Place doors from the catalog* (on page 284)
- *Place windows from the catalog* (on page 285)
- *Edit equipment properties* (on page 286)
- *Edit equipment relationships* (on page 287)

Place equipment from the catalog

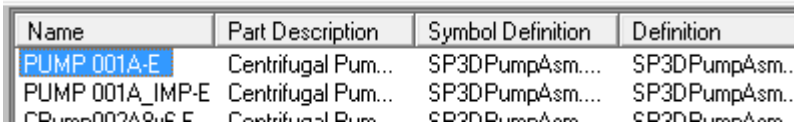
1. Click **Place Equipment** on the vertical toolbar.

   *The **Select Equipment** dialog box displays.*

2. Locate the necessary equipment type using the tree view. Expand the nodes for the general type of equipment that you need, continuing until a list of available parts appears in the catalog window.



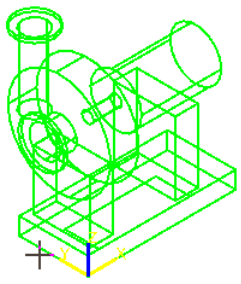
3. In the list view, select the equipment object.



4. Click **OK** on the **Select Equipment** dialog box.

*The **Equipment Properties** dialog box displays so that you can define properties for the new object.*

5. Define properties as needed, and then click **OK** to return to the workspace and place the equipment. For more information, see *Equipment Properties Dialog Box* (on page 287).
6. Click in a graphic view to select the mounting surface and approximate position for the object.



TIP Press the left and right arrow keys to rotate the equipment by 90-degree increments at any time during the placement of the equipment. Press the up arrow key to scroll through the three possible axes of rotation. Press the down arrow key to scroll to each datum point.

7. Using the **Place Equipment** ribbon, do any of the following, if needed:
   a. Set an offset for the relationship by specifying the distance in the **Offset** box.
   b. Click **Properties** on the ribbon, and type any necessary property information in the grid provided on the **Equipment Properties** dialog box.
   c. Add or change a positioning relationship by selecting it from the **Relationship** list on the **Place Equipment** ribbon.
   d. Select a system with which to associate the new object in the **System** box.

NOTES If you intend to use an object for routing cable, you must not model that object as equipment. For example, a pull box should be created in the catalog data as a conduit fitting, not a piece of equipment. Otherwise, the object will render conduit networks useless for cable routing every time it is used.

Set positioning relationships for equipment

1. Select the equipment for which you need to define a relationship.
2. In the **Relationship List** box on the **Place Equipment** ribbon, select <**New Relationship**>.
3. Select one of the available positioning relationships in the **Positioning Relationships** list box, and follow the status bar prompts.

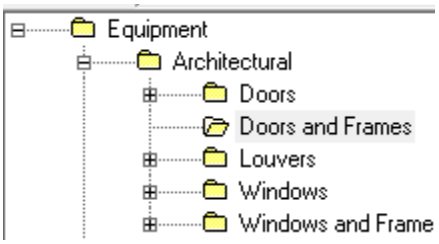
   NOTE Depending on the type of positioning relationship defined in the previous step, surfaces or points need to be defined to finish configuring the relationship. For more information, see Positioning Relationships.

Place doors from the catalog

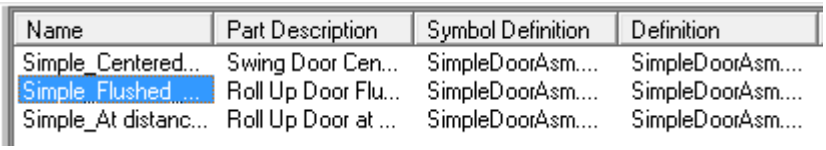
1. Click **Place Equipment** on the vertical toolbar.

   *The **Select Equipment** dialog box displays.*

2. Locate the door to place under the Equipment > Architectural node.



3. In the list view, select the door to place.



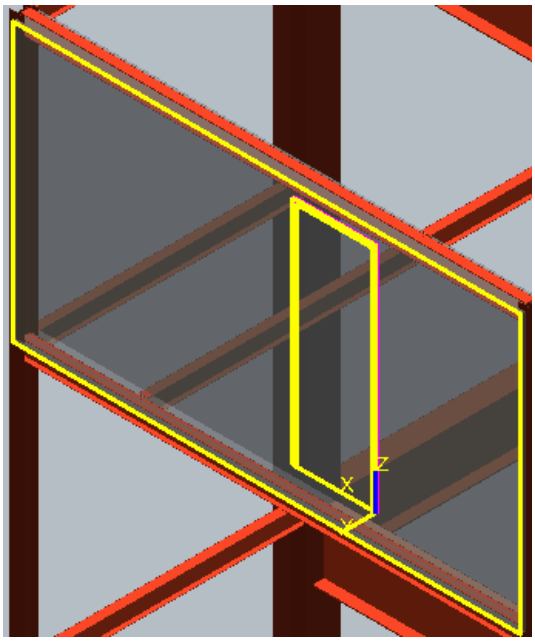
Name	Part Description	Symbol Definition	Definition
Simple_Centered...	Swing Door Cen...	SimpleDoorAsm....	SimpleDoorAsm....
Simple_Flushed_...	Roll Up Door Flu...	SimpleDoorAsm....	SimpleDoorAsm....
Simple_At distanc...	Roll Up Door at ...	SimpleDoorAsm....	SimpleDoorAsm....

4. Click **OK** on the **Select Equipment** dialog box.

   *The **Equipment Properties** dialog box displays so that you can define properties for the new object.*

5. Define properties as needed, and then click **OK** to return to the workspace and place the equipment. For more information, see *Equipment Properties Dialog Box* (on page 287).

6. Click in a graphic view to select the mounting surface and approximate position for the object.



TIP Press the left and right arrow keys to rotate the door by 90-degree increments at any time during the placement of the door. Press the up arrow key to scroll through the three possible axes of rotation.

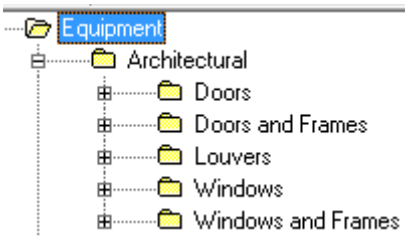
7. Using the **Place Equipment** ribbon, do any of the following, if needed:
   a. Set an offset for the relationship by specifying the distance in the **Offset** box.
   b. Click **Properties** on the ribbon, and type any necessary property information in the grid provided on the **Equipment Properties** dialog box.
   c. Add or change a positioning relationship by selecting it from the **Relationship** list on the **Place Equipment** ribbon.
   d. Select a system with which to associate the new object in the **System** box. We recommend that you place the door in the same system as the parent wall.

Place windows from the catalog

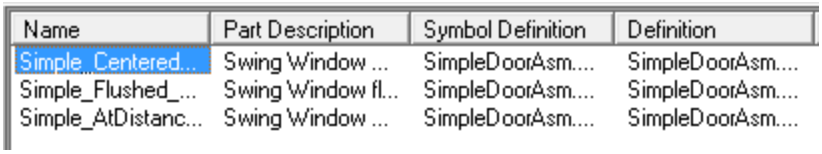
1. Click **Place Equipment** on the vertical toolbar.

   *The **Select Equipment** dialog box displays.*

2. Locate the window to place under the Equipment > Architectural > Windows node.



3. In the list view, select the window part to place.

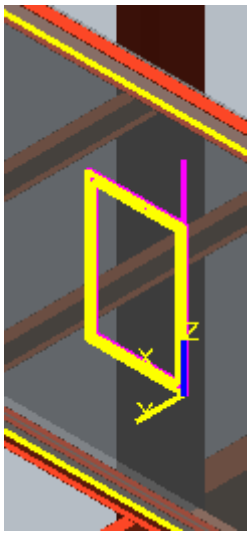


4. Click **OK** on the **Select Equipment** dialog box.

   *The **Equipment Properties** dialog box displays so that you can define properties for the new object.*

5. Define properties as needed, and then click **OK** to return to the workspace and place the equipment. For more information, see *Equipment Properties Dialog Box* (on page 287).

6. Click in a graphic view to select the mounting surface and approximate position for the object.



TIP Press the left and right arrow keys to rotate the window by 90-degree increments at any time during the placement of the window. Press the up arrow key to scroll through the three possible axes of rotation.

7. Using the **Place Equipment** ribbon, do any of the following, if needed:
   a. Set an offset for the relationship by specifying the distance in the **Offset** box.
   b. Click **Properties** on the ribbon, and type any necessary property information in the grid provided on the **Equipment Properties** dialog box.
   c. Add or change a positioning relationship by selecting it from the **Relationship** list on the **Place Equipment** ribbon.
   d. Select a system with which to associate the new object in the **System** box. We recommend that you place the window in the same system as the parent wall.

Edit equipment properties

1. Select the equipment object that you need to modify.
2. Click **Equipment Properties** on the horizontal ribbon.

   *The **Equipment Properties** dialog box displays.*

   TIP You can also access the **Equipment Properties** dialog box by selecting **Properties** from the **Edit** menu, or by right-clicking the selected object and choosing **Properties** from the short-cut menu.
3. Choose the appropriate dialog box tabs, and modify the properties as needed.
4. Click **OK** to save your changes, and return to the workspace.

   TIP You can also select **Apply** to put the changes into effect and continue working in the **Equipment Properties** dialog box.

NOTES

All occurrence and definition properties for an equipment object are defined by the Microsoft Excel® workbook named **Equipment.xls** in the reference data. For information on adding or editing different types of properties, refer to the *Reference Data Guide,* accessible from the **Help** > **Printable Guides** command.

- If more than one piece of equipment is selected, the **Occurrence** tab displays only the common occurrence properties of the objects.
- The **Definition**, **Connections**, **Weight & CG**, and **Relationship** tabs are read-only. You cannot edit this information from the **Equipment Properties** dialog box.
- If the equipment model was created using Solid Edge, then any changes that affect the shape or size of the equipment model require that you have Solid Edge running on your computer.

Edit equipment relationships

1. Select the equipment to modify.

   TIP To edit the properties of a designed equipment object for which geometry has yet to be defined, you must select the object from the system hierarchy in the **Workspace Explorer**.

2. Specify the relationship in the **Relationship List** box.
3. Change the type of relationship using the **Positioning Relationship** box as needed.

   Set positioning relationships for equipment (on page 283)
   Set positioning relationships for designed equipment (on page 302)

   NOTE You can remove relationships by clicking **Delete Relationship** on the ribbon. This step is often necessary when moving previously constrained equipment. After a relationship is deleted, no connectivity remains between the two pieces of equipment during further design operations.

Equipment Properties Dialog Box

Displays equipment properties for review and editing.

See Also

Occurrence Tab (Equipment Properties Dialog Box) (on page 288)
Definition Tab (Equipment Properties Dialog Box) (on page 296)
Connection Tab (on page 297)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)
Edit Equipment Properties (on page 286)
Edit Equipment Relationships (on page 287)

Occurrence Tab (Equipment Properties Dialog Box)

Displays all the editable instance-specific information about the selected equipment object. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If more than one equipment object is selected, only the common occurrence properties for the selected objects appear on the tab.

When viewing properties for a single equipment object, the following properties appear. More properties may appear depending on what you defined in the reference data. For more information about occurrences defined in the reference data, see the *Equipment and Furnishings Reference Data Guide* available with the **Help** > **Printable Guides** command in the software.

NOTE Because equipment properties are customizable in the equipment and furnishings reference data, only the properties that are required by the software are documented.

Show Dimensional Legend

Displays the bitmap image associated with the equipment object if it has been defined in the reference data. The image displays in a separate window.

Category

Select the properties to view, modify, or define. Equipment properties are divided into several different categories: **Standard**, **Position and Orientation**, **Insulation and Tracing**, **Weight and CG**, **Equipment Dimension**, **Fabrication and Construction**, **Surface Treatment and Coating**, and **Responsibility**.

Standard

Name

Displays the name of the equipment object. The equipment name is based on the **Name Rule** selection. If you type a name in this field, the **Name Rule** property updates to **User Defined**.

Name Rule

Specify the naming rule to use to name this equipment object. You can select one of the listed rules or select **User Defined** to specify the equipment name yourself in the **Name** box.

Description

Specifies a description for the object.

Equipment

Select the system to which the object belongs. By default, the model is the parent system for the object. If a parent system is selected when you start the command, **System** is automatically updated with the selected system. If multiple systems are selected when you start the command, **System** is automatically updated with the last used system.

Reporting Requirements

Specifies the reporting requirement for the object. To change the options on the list, edit the **Reporting Type** select list in Catalog.

Reporting Type

Specifies the type of reporting. To change the options on the list, edit the **Reporting Type** select list in Catalog.

Behavior Controlled by User

Indicates whether or not you can delete the object. Select **False** to prevent anyone from deleting the object from the model.

Correlation Status

Specifies whether or not the object has been correlated to an object in a P&ID. The list is defined by the EFWCorrelationStatus select list.

Correlation Basis

Specifies whether or not the object is correlated to a P&ID object. The list is defined by the EFWCorrelationBasis select list.

Correlate Object indicates that the object has a correlating object in a P&ID.

No correlation is required indicates that the object does not have a correlating object in a P&ID.

Correlation Approval Status

Specifies whether or not the object is approved with discrepancies in the three-dimensional data compared with design basis data. The list is defined by the EFWCorrelationApprovalStatus sheet in the AllCodelist workbook.

Topology mismatch approved indicates that the object topology mismatch can be ignored.

Data and Topology mismatches approved indicates that the object data and topology mismatches can be ignored.

None indicates that you do not approve a mismatch.

Position and Orientation

East

Displays the distance of the connection point from the active coordinate system origin in the east direction.

North

Displays the distance of the connection point from the active coordinate system origin in the north direction.

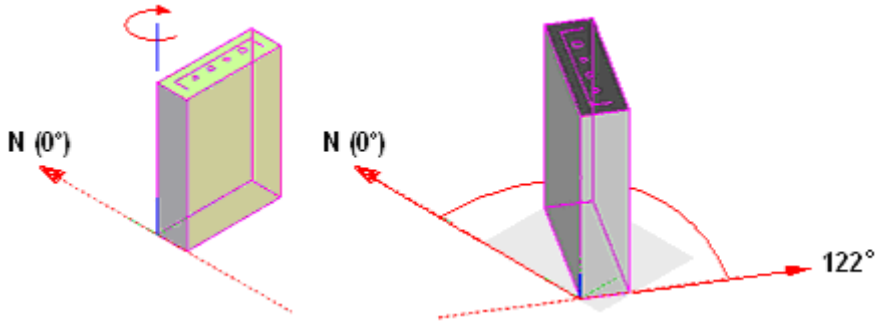
Elevation

Displays the distance of the connection point above or below the active coordinate system origin.

Bearing

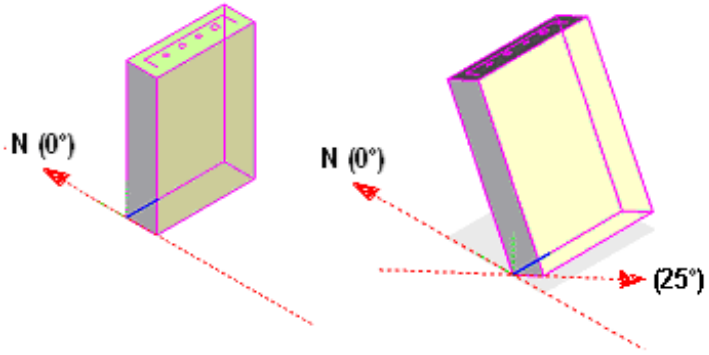
Specifies the bearing angle for the object. The **Bearing** angle is measured between the local x-axis of the object and the Y-axis (North) of the global coordinate system in the XY-plane. The local x-axis is the default axis of primary symmetry for all symbols in the catalog. The bearing measurement direction is clockwise from the active coordinate system North looking in the negative active coordinate system direction; that is, down from 0 to 360 degrees. You can enter negative bearing angles, but the software automatically converts them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the bearing

measure displays 0 degrees.



Pitch

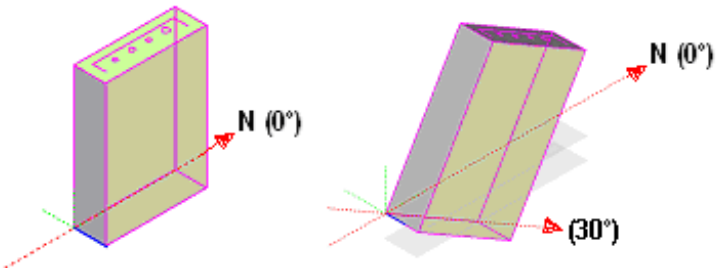
Specifies the pitch angle for the shape. The **Pitch** angle measures between the x-axis of the object and the X-axis of the global coordinate system in the XZ-plane. This option sets the reference in the model to a line that is the intersection of the vertical plane through the X-axis of the local coordinate system and the active coordinate system horizontal plane. The angle is measured in the positive direction from the horizontal plane in the active coordinate system up direction regardless of the current bearing. Another way to describe pitch is the rotation of the object about its y-axis. Pitch angles are limited to between -90 degrees and +90 degrees, with 0 indicating horizontal.



Roll

Specifies the roll angle for the object. The **Roll** angle measures between the local z-axis of the object and the Z-axis of the global coordinate in the YZ-plane. Another way to describe roll is the rotation of the object about its x-axis. This option sets the reference in the model to a line that is perpendicular to the local coordinate system x-axis and in the horizontal plane. Roll angles are measured clockwise from horizontal to the y-axis of the local coordinate system. The roll angle is between 0 and 360 degrees. You can enter negative roll angles, but the software automatically converts them to the positive equivalents. If the pitch is set to

+/- 90 degrees, then the reference in the model is the North axis.



Insulation and Tracing

Is Insulated

Indicates whether or not the object is insulated.

Thickness

Specifies the thickness of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Thickness** box. However, if **Insulation Specification** is manually defined, select the insulation thickness from those values allowed for the material in the reference data. If the value in the **Insulation Specification** field is a selected reference data insulation specification, the software reads the thickness from the insulation specification from the reference data and displays it here.

Requirement

Specifies the requirement of the insulation. To change the options on the list, edit the **Insulation Purpose** select list in Catalog.

Insulation Type

Specifies the type of the insulation. To change the options on the list, edit the **Insulation Purpose** select list in Catalog.

Insulation Purpose

Displays the purpose of the insulation. If **Insulation Specification** is set to use the insulation defined by the equipment object, the information in this box cannot be changed. However, if **Insulation Specification** is manually defined, select the purpose of the insulation in this box. Available purposes are read from the reference data.

Material

Specifies the material of the insulation. If **Insulation Specification** is set to use the insulation defined by the equipment object, this field cannot be changed. However, if **Insulation Specification** is manually defined, select the insulation material from those available from the reference data.

The list contains the last five insulation materials selected. Click **More** to browse the catalog for the insulation material to use. If you set this field to **Not Insulated**, the **Insulation Thickness** field is disabled. If you set this field to **Use Run Material**, the along leg feature inherits the insulation material from the duct run.

To change the options on the list, edit the **Insulation Material** select list in Catalog.

Operating Temperature

Specifies the operating temperature.

Insulation Surface Area

Specifies the measurement of the surface area of insulation.

Heat Tracing Requirement

Indicates whether or not the equipment is heat-traced. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Type

Specifies the type of heat-tracing. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Medium

Specifies the heat-tracing medium to apply to the object. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Medium Temperature

Specifies the temperature of the heat-tracing medium. Include the unit of measurement of temperature (**K** for Kelvin, **F** for Fahrenheit, or **C** for Celsius, for example).

Weight & CG

Displays the center-of-gravity and the weight of the selected equipment objects. The center-of-gravity locations are displayed in global system coordinates along the X-, Y-, and Z-axes. The software includes the insulation weight in the calculated weight. If you specify the weight yourself, you must include the insulation weight value.

Dry Weight

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

NOTE For equipment, the **Weight and CG** property **Wet Weight** is the sum of **Dry Weight** and **Water Weight**. The dry weight and water weight values are catalog properties entered on the part sheet for the equipment.

Dry CGX

Specifies the X-axis location of the dry center-of-gravity.

Dry CGY

Specifies the Y-axis location of the dry center-of-gravity.

Dry CGZ

Specifies the Z-axis location of the dry center-of-gravity.

Wet CGX

Specifies the X-axis location of the wet center-of-gravity.

Wet CGY

Specifies the Y-axis location of the wet center-of-gravity.

Wet CGZ

Specifies the Z-axis location of the wet center-of-gravity.

Equipment Dimension

Property

Displays the name of the property. The properties that appear are dependent on the equipment type selected. For example, the properties displayed for an equipment object based on a pump are different from those based on a cooler. For more information on equipment part properties, see the *Equipment and Furnishings Reference Data Guide*, available with the **Help > Printable Guides** command within the **Equipment and Furnishings** task.

Value

Displays the value of the corresponding property.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Surface Treatment and Coating

Exterior Coating Requirement

Specifies the coating requirement for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Exterior Coating Area

Specifies the area of the coating for the object.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

See Also

Equipment Properties Dialog Box (on page 287)

Select Insulation Material Dialog Box

Specifies the insulation materials for a selected object. By browsing through the part hierarchy, you can find and select an insulation material in the Catalog database.

 **Save**

This option is unavailable.

Cut

This option is unavailable.

Copy

This option is unavailable.

Paste

This option is unavailable.

Delete

This option is unavailable.

Undo

This option is unavailable.

Insert Row

This option is unavailable.

Move Up

This option is unavailable.

Move Down

This option is unavailable.

Properties

Displays the properties of the selected object. Because you cannot modify any properties until the object is placed, all properties on the **Properties** dialog box are read-only.

Preview

Displays a bitmap symbol of the selected object. The image file must be assigned to the object in the catalog reference data.

Filter

This option is unavailable.

Sort

This option is unavailable.

Customize Current View

This option is unavailable.

List View

Sets the dialog box to display insulation thickness values in a list view.

Grid View

Sets the dialog box to display insulation thickness values in a spreadsheet-style grid view.

Back

Returns you to the previously selected insulation material folder. Use this command to navigate through the hierarchy to the specific insulation material you need.

Forward

Sends you to the last selected insulation material folder that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific insulation material you need.

Up One Level

Brings up the next highest level of the hierarchy. Use this command to navigate through the hierarchy to the specific material you need.

Check Data

This option is unavailable.

CheckData Ignored Inconsistencies

This option is unavailable.

Smart 3D Help

This option is unavailable.

Address

Sets the current location within the hierarchy of available insulation materials. The dropdown box lists the folders you have visited.

Definition Tab (Equipment Properties Dialog Box)

Displays the part information for the equipment object, the properties and their values, as defined in the reference data. If more than one equipment object is selected, only the common properties for the selected objects display on the tab. For more information about the information defined in the reference data, see the *Equipment and Furnishings Reference Data Guide,* available from the **Help > Printable Guides** command in the software.

Category

Select the properties to review. Equipment information is divided into several different categories, such as **Standard**, **Equipment Specification**, **Equipment Support**, **Nozzle Length**, and **Equipment Dimensions**.

IMPORTANT Not all categories are available for all equipment objects.

Property

Displays the name of the property. The properties that appear are dependent on the equipment type selected. For example, the properties displayed for an equipment object based on a pump are different from those based on a cooler. For more information on equipment part properties, see *the Equipment and Furnishings Reference Data Guide*, available with the **Help** > **Printable Guides** command within the Equipment and Furnishings task.

Value

Displays the value of the corresponding property.

Connection Tab

Displays information on the connection points of a piece of equipment, including piping, electrical, foundation, or HVAC connections, and Reference 3D objects. If more than one equipment object is selected, only the common properties of the connections for the selected objects display on the tab. For more information about the information defined in the reference data, see the *Equipment and Furnishings Reference Data Guide*, available from the **Help** > **Printable Guides** command in the software.

Connector

Select the connector for which to view properties. With the exception of Reference 3D object connectors, the connectors display in alphabetical order. When you select a connector from the list, the graphic object associated with the connection, if one exists, highlights in the graphic view for further visual confirmation of the port selected.

Property

Lists all the properties of the selected connection. These properties vary depending on the type of connection selected.

Value

Displays the value of the corresponding property.

Insulation Tab

Displays insulation properties that have been defined for the selected equipment.

Insulation Specification

Specifies whether or not the selected equipment object is insulated. The options on this dialog box remain disabled unless **User Defined** is selected from the dropdown list.

Purpose

Specifies the insulation purpose for the selected equipment object.

Material

Displays the **Select Insulation Material** dialog box from which you can specify an insulation material. For more information, see *Select Insulation Material Dialog Box* (on page 294).

Thickness

Defines the thickness of the specified insulation material. This option remains disabled until a material is selected from the **Select Insulation Material** dialog box. By default, the smallest thickness value of the selected material is displayed. An alternative value may be selected from the dropdown list.

See Also

Designed Equipment Properties Dialog Box (on page 304)
Equipment Properties Dialog Box (on page 287)

SECTION 18

Place Designed Equipment

Places equipment types that have been defined in the reference data. The properties of the equipment type you select are inherited by the designed equipment part. With the **Place Designed Equipment** command, you can define an equipment type, a system parent, name, and other property values as appropriate for the equipment type. Equipment position and orientation is further defined by mating, aligning, or connecting equipment graphics to reference graphics in the workspace, or by moving or rotating the equipment.

The software's designed equipment component modeling capabilities allow you to build an equipment definition in the Model database by combining basic shapes, ports (nozzle, foundation, electrical, and so forth) and properties defined in the reference data. Designed equipment objects can be built entirely out of primitive geometric shapes, designed equipment components, or an existing equipment component part can be placed from the catalog to enhance the designed equipment object. For example, you can use the **Place Designed Equipment** command to create a designed equipment object in the model, add an agitator (using the **Place Shape** command), and then add nozzle primitives (using the **Place Nozzle** command) to customize the designed equipment. You can also place a nozzle using a nozzle defined in the P&ID design basis (if P&ID design basis data is available).

Some equipment shapes can be modeled more easily using solid modeling software. In addition to using the primitive shapes delivered with the software, you can import graphics from an SAT file as the geometry for the designed equipment shape. After the geometry is defined, you can then place ports to define distributed connections to the designed equipment.

As a further enhancement, the software provides a prismatic shape feature so that you can design uniquely customized shapes in the model. The place prismatic shape feature is made up of two distinct processes:

- Define a path.
- Define a cross section, or profile, to project along the path.

The ultimate goal of these two processes is placement of a shape in the model as part of a designed equipment object.

After a path and cross-section have been defined, the cross- section is then projected along the path to create a shape with the same properties as other equipment shapes.

NOTE The place prismatic shape feature is available on the **Occurrence** tab of the **Shape Properties** dialog box.

When you create a designed equipment object and add an equipment component object or shape, the default surface on the equipment component object or shape as defined in Visual Basic is automatically used to establish a mate or align relationship to the other surface or reference element you select during initial placement. When modifying a shape to create a new mate or align relationship, the default surface is not automatically used. You must interactively select the surface. For example, you might define the bottom of a pump as the default surface so that the software mates that surface to the floor. If the software cannot find a suitable mating surface for the equipment, the equipment object is placed in free space, pending additional design of the structures or other reference elements.

When creating a designed equipment object, you can provide an optional offset distance from the surface. The default offset distance for any new piece of equipment is zero or the last offset used in the current session. The software maintains the offset relationship between the default surface and the surface or reference element in the workspace in the event of any changes to their position. For example, if a slab is lowered, then the equipment follows the slab, maintaining the offset. You can further define the equipment orientation by mating, aligning, or connecting equipment to other objects in the model, or by moving or rotating the designed equipment. For more information on positioning relationships, see Positioning Relationships.

Place Designed Equipment Ribbon

Sets options for adding designed equipment to your model. This ribbon appears automatically after you select the **Place Designed Equipment** command and then select an equipment type. Until a shape has been added to the designed equipment, only the **Equipment Properties**, **Equipment Name**, and **Active System** controls are available.

TIPS

- To find out the name of an option on the ribbon, pause the pointer over an option and read the ToolTip.
- To add a shape to the designed equipment and enable the remaining ribbon controls, select **Place Shape**.

Equipment Properties

Edits the occurrence properties and review the static properties of an existing piece of equipment. Equipment properties can be edited only after the designed equipment is placed in the model.

User Defined Form Definition

Displays **User Defined Form** for the particular equipment if it is defined in the catalog. See the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

Relationship List

Lists all relationships for the selected designed equipment and provides an option for creating a new relationship if the equipment is not already fully constrained. An equipment object is fully constrained when it has sufficient defined relationships to prevent movement or rotation of the object along all three coordinate axes. This control is only available after a shape has been added to the designed equipment.

Positioning Relationships

Displays the available options for types of positioning relationships: Mate, Align, Connect Minimum Distance, Mate to Tangent Plane, and Parallel. Some options may not be available for all designed equipment types. See Positioning Relationships for more information.

Delete Relationship

Removes the selected relationship from the equipment model and the database. Using the **Relationships List** box, select a previously existing relationship for the designed equipment, and click **Delete Relationship**. You can use this command only when modifying existing designed equipment. This control is available only after a shape has been added to the designed equipment.

Equipment Reference

Prompts you for the reference on the designed equipment to be placed that will be affected by the positioning relationship. In all cases, the part that you select in this step moves to create the relationship, and the part chosen in the **Second Part Reference** step remains fixed.

Second Part Reference

Prompts you for the reference on the equipment object or reference element already in the model that will be affected by the positioning relationship. After you select the reference, the software repositions the first designed equipment object chosen with respect to the second object selected in the definition of the relationship. This control is available only after a shape has been added to the designed equipment.

Offset

Defines the offset distance for a mate or align relationship. Offsets are disabled when establishing a connect relationship. You can adjust this value after initial equipment placement if needed. This control is available only after a shape has been added to the designed equipment.

Name

Displays the designed equipment name, as dictated by your predefined name rules, and accepts changes to that name.

System

Specifies the system with which to associate the selected equipment. The default system is the Model itself.

Select Equipment Type Dialog Box

Selects the type of designed equipment to be created. This dialog box appears automatically when you click the **Place Designed Equipment** command. By browsing through the part hierarchy, you can find any piece of equipment in the Catalog database. The resulting designed equipment inherits the properties of the existing equipment type you select from the dialog box. After you select an equipment type and click **OK**, the **Properties** dialog box appears so you can set properties for the new equipment. After you click **OK** on the **Properties** dialog box, the software returns you to the model so you can finish placing the object.

Back

Returns you to the previously selected equipment type or node. Use this command to navigate through the equipment hierarchy to the specific type you need.

Forward

Sends you to the last selected equipment type or node that you moved away from by using the **Back** button. Use this command to navigate through the equipment hierarchy to the specific type you need.

Up One Level

Brings up the next highest level of the Equipment catalog hierarchy. Use this command to navigate through the equipment hierarchy to the specific type you need.

Properties

Displays the properties of the selected object. Because you cannot modify any properties until the equipment is placed, all properties on the **Properties** dialog box are read-only.

Preview

Displays a bitmap symbol of the selected equipment. The image file must be assigned to the equipment in the catalog reference data.

List View

Sets the dialog box to display equipment in a list view.

Grid View

Sets the dialog box to display equipment in a spreadsheet-style grid view.

Address

Specifies your exact location within the displayed hierarchy.

What do you want to do?

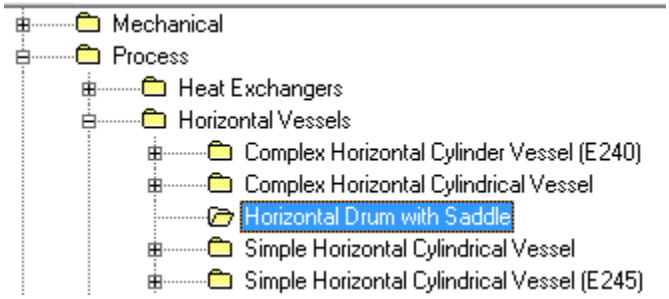
- *Place designed equipment* (on page 301)
- *Set positioning relationships for designed equipment* (on page 302)
- *Edit designed equipment properties* (on page 302)
- *Edit designed equipment reference coordinate system* (on page 303)

Place designed equipment

1. Click **Place Designed Equipment** on the vertical toolbar.

   *The **Select Equipment Type** dialog box displays.*

2. Select the designed equipment type to create.



IMPORTANT The **Select Equipment Type** dialog box displays the existing equipment classifications as defined in the Catalog database. The resulting designed equipment object inherits the properties of the equipment type you select.

3. Click **OK**.

   *A definition of the new designed equipment object is created in the Model database and displays in the system hierarchy in the **Workplace Explorer**.*

*The **Designed Equipment Properties** dialog box displays.*

4. Edit the designed equipment properties as needed, and click **OK**.

   The software returns you to the model to complete the placement of the designed equipment.

5. Click in the graphic view to define the origin of the equipment.

   TIP You can press the down arrow key to scroll to each data point in the equipment.

6. If necessary, use the **Place Designed Equipment** ribbon to do the following:

   - Select a system with which to associate the new designed equipment object from the options in the **System** box.
   - Type a name for the new designed equipment object in the **Name** box.

   NOTE Although the new designed equipment object is added to the **Workspace Explorer**, you must use **Place Shape** to create geometry for the object. For more information, see *Place Shape* (on page 328).

Set positioning relationships for designed equipment

1. Select the designed equipment object for which to define a relationship.
2. In the **Relationships List** box on the **Place Designed Equipment** ribbon, choose <**New Relationship**>.
3. Select one of the available positioning relationships in the **Positioning Relationships List** box, and follow the on-screen prompts. For more information, see Positioning Relationships.

   NOTE Depending on the type of positioning relationship defined in the previous step, surfaces or points need to be defined to finish configuring the relationship.

Edit designed equipment properties

1. On the horizontal ribbon, set the **Locate Filter** box to **Equipment**, and then select an equipment object.

   TIPS

   - If the **Locate Filter** box is not displayed on the horizontal ribbon, click **Select** on the vertical toolbar.
   - To edit the properties of a designed equipment object for which geometry has yet to be defined, you must select the object from the system hierarchy in the **Workspace Explorer**.

2. Click **Properties** on the horizontal ribbon.

   *The **Properties** dialog box displays.*

   TIP You can also access the **Properties** dialog box by selecting **Properties** from the **Edit** menu, or by right-clicking an equipment object and choosing **Properties** from the short-cut menu.

3. Click the appropriate dialog box tabs, and modify the properties as needed.

4. Click **OK** to save your changes, and return to the workspace.

   TIP You can also select **Apply** to put the changes into effect and continue working in the **Properties** dialog box.

NOTES

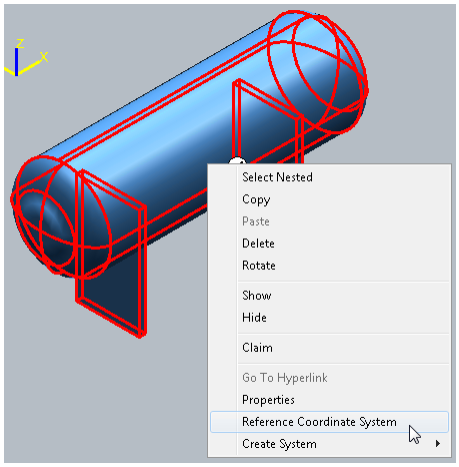
- Occurrence and definition properties for an equipment object are defined by the Excel workbook named **Equipment.xls** in the reference data. For information on adding or editing different types of properties, refer to the *Reference Data Guide,* accessible from the **Help > Printable Guides** command in the software.
- If more than one piece of equipment is selected, the **Occurrence** tab displays only the common occurrence properties of the items.
- The **Definition**, **Connections**, **Weight & CG**, and **Relationship** tabs are read-only. The properties and values displayed on these tabs are defined in the reference data. For more information, see the *Equipment and Furnishings Reference Data Guide,* accessible from the **Help > Printable Guides** command in the software.
- If the equipment model was created using Solid Edge, then any changes that affect the shape or size of the equipment model require that you have Solid Edge running on your computer.

Edit designed equipment reference coordinate system

1. On the horizontal ribbon, set the **Locate Filter** box to **Equipment**, and then select a designed equipment object.

   TIP If the **Locate Filter** box is not displayed on the horizontal ribbon, click **Select** on the vertical toolbar.

2. Right-click the selected equipment object, and click **Reference coordinate system.**



3. Click **Move** to move the reference coordinate system.

   The control point of the designed equipment moves with the reference coordinate system.

Designed Equipment Properties Dialog Box

Displays designed equipment properties for review and editing.

See Also

Occurrence Tab (Designed Equipment Properties Dialog Box) (on page 304)
Definition Tab (Equipment Properties Dialog Box) (on page 296)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

Occurrence Tab (Designed Equipment Properties Dialog Box)

Displays all the editable instance-specific information about the selected designed equipment object. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If more than one equipment object is selected, common occurrence properties for the selected objects appear on the tab.

When viewing properties for a single equipment object, the following properties appear. More properties may appear depending on what you defined in the reference data. For more information about occurrences defined in the reference data, see the *Equipment and Furnishings Reference Data Guide,* available from the **Help** > **Printable Guides** command in the software.

NOTE Because equipment properties are customizable in the equipment and furnishings reference data, only the properties that are required by the software are documented.

Display Equipment Preview

This button is disabled for designed equipment.

Category

Select the properties to view, modify, or define. Equipment properties are divided into several different categories: Standard, Insulation and Tracing, Weight and CG, Fabrication and Construction, Surface Treatment and Coating, Position and Orientation, and Responsibility.

Standard

Name

Specifies the name of the object. If a **Name Rule** is specified, then the software uses that rule to determine this name. If the **Name Rule** value is **User Defined**, then you must type a name in this box.

Name Rule

Displays the available name rules for the selected object. Specify the naming rule to use to name the object. You can select one of the listed rules, or you can select **User Defined** to specify the name yourself in the **(Name)** box. **Description**

Specifies a description for the object.

System

Select the system to which the object belongs. By default, the model is the parent system for the object. If a parent system is selected when you start the command, **System** is automatically updated with the selected system. If multiple systems are selected when you start the command, **System** is automatically updated with the last used system.

Reporting Requirements

Specifies the reporting requirement for the object. To change the options on the list, edit the **Reporting Type** select list in Catalog.

Reporting Type

Specifies the type of reporting. To change the options on the list, edit the **Reporting Type** select list in Catalog.

Room Number

Specifies the room number associated with the designed equipment.

Correlation Status

Specifies whether or not the object has been correlated to an object in a P&ID. The list is defined by the EFWCorrelationStatus select list.

Correlation Basis

Specifies whether or not the object is correlated to a P&ID object. The list is defined by the EFWCorrelationBasis select list.

Correlate Object indicates that the object has a correlating object in a P&ID.

No correlation is required indicates that the object does not have a correlating object in a P&ID.

Correlation Approval Status

Specifies whether or not the object is approved with discrepancies in the three-dimensional data compared with design basis data. The list is defined by the EFWCorrelationApprovalStatus sheet in the AllCodelist workbook.

Topology mismatch approved indicates that the object topology mismatch can be ignored.

Data and Topology mismatches approved indicates that the object data and topology mismatches can be ignored.

None indicates that you do not approve a mismatch.

Position and Orientation

East

Displays the distance of the connection point from the active coordinate system origin in the east direction.

North

Displays the distance of the connection point from the active coordinate system origin in the north direction.

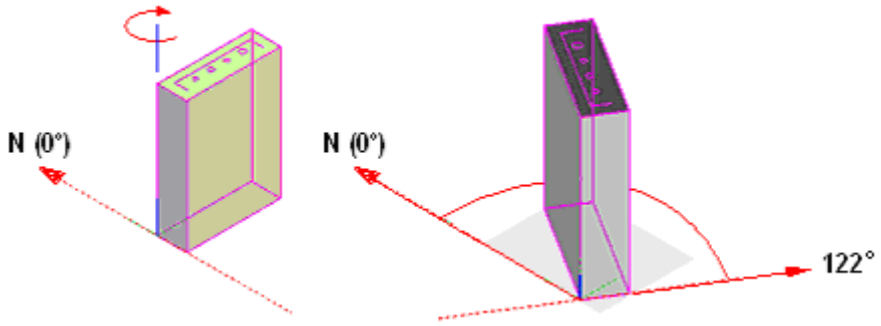
Elevation

Displays the distance of the connection point above or below the active coordinate system

origin.

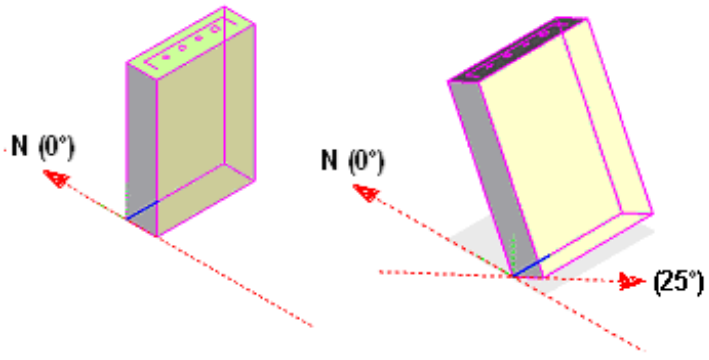
Bearing

Specifies the bearing angle for the object. The **Bearing** angle is measured between the local x-axis of the object and the Y-axis (North) of the global coordinate system in the XY-plane. The local x-axis is the default axis of primary symmetry for all symbols in the catalog. The bearing measurement direction is clockwise from the active coordinate system North looking in the negative active coordinate system direction; that is, down from 0 to 360 degrees. You can enter negative bearing angles, but the software automatically converts them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the bearing measure displays 0 degrees.



Pitch

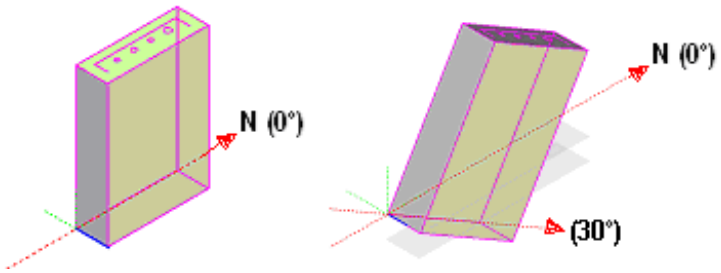
Specifies the pitch angle for the shape. The **Pitch** angle measures between the x-axis of the object and the X-axis of the global coordinate system in the XZ-plane. This option sets the reference in the model to a line that is the intersection of the vertical plane through the X-axis of the local coordinate system and the active coordinate system horizontal plane. The angle is measured in the positive direction from the horizontal plane in the active coordinate system up direction regardless of the current bearing. Another way to describe pitch is the rotation of the object about its y-axis. Pitch angles are limited to between -90 degrees and +90 degrees, with 0 indicating horizontal.



Roll

Specifies the roll angle for the object. The **Roll** angle measures between the local z-axis of the object and the Z-axis of the global coordinate in the YZ-plane. Another way to describe roll is the rotation of the object about its x-axis. This option sets the reference in the model to a line that is perpendicular to the local coordinate system x-axis and in the horizontal plane. Roll angles are measured clockwise from horizontal to the y-axis of the local coordinate

system. The roll angle is between 0 and 360 degrees. You can enter negative roll angles, but the software automatically converts them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the reference in the model is the North axis.



Insulation and Tracing

Is Insulated

Indicates whether or not the object is insulated.

Thickness

Specifies the thickness of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Thickness** box. However, if **Insulation Specification** is manually defined, select the insulation thickness from those values allowed for the material in the reference data. If the value in the **Insulation Specification** field is a selected reference data insulation specification, the software reads the thickness from the insulation specification from the reference data and displays it here.

Insulation Purpose

Displays the purpose of the insulation. If **Insulation Specification** is set to use the insulation defined by the equipment object, the information in this box cannot be changed. However, if **Insulation Specification** is manually defined, select the purpose of the insulation in this box. Available purposes are read from the reference data.

Insulation Type

Specifies the type of the insulation. To change the options on the list, edit the **Insulation Purpose** select list in Catalog.

Material

Specifies the material of the insulation. If you selected **Not Insulated** in the **Insulation Specification** box, you cannot change the information in the **Insulation Material** box. However, if **Insulation Specification** is manually defined, you can select the insulation material to use. If **Insulation Specification** is a selected reference data insulation specification, the software reads the material from the insulation specification as defined in the reference data and displays it here.

Select **More** to select an insulation material from the Catalog. To change the options on the list, edit the **Insulation Material** select list in Catalog.

Operating Temperature

Specifies the operating temperature.

Insulation Surface Area

Specifies the measurement of the surface area of insulation.

Heat Tracing Requirement

Indicates whether or not the equipment is heat-traced. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Type

Specifies the type of heat-tracing. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Medium

Specifies the heat-tracing medium to apply to the object. To change the options on the list, edit the **Heat Tracing Medium** select list in Catalog.

Heat Tracing Medium Temperature

Specifies the temperature of the heat-tracing medium. Include the unit of measurement of temperature (**K** for Kelvin, **F** for Fahrenheit, or **C** for Celsius, for example).

Weight & CG

Dry Weight

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

NOTE For equipment, the **Weight and CG** property **Wet Weight** is the sum of **Dry Weight** and **Water Weight**. The dry weight and water weight values are catalog properties entered on the part sheet for the equipment.

Dry CGX

Specifies the X-axis location of the dry center-of-gravity.

Dry CGY

Specifies the Y-axis location of the dry center-of-gravity.

Dry CGZ

Specifies the Z-axis location of the dry center-of-gravity.

Wet CGX

Specifies the X-axis location of the wet center-of-gravity.

Wet CGY

Specifies the Y-axis location of the wet center-of-gravity.

Wet CGZ

Specifies the Z-axis location of the wet center-of-gravity.

Dry WCG Origin

Specifies the way in which the dry weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the dry weight center-of-gravity location relative to the active coordinate system.

Wet WCG Origin

Specifies the way in which the wet weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the wet weight center-of-gravity location relative to the active coordinate system.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Surface Treatment and Coating

Exterior Coating Requirement

Specifies the coating requirement for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Exterior Coating Area

Specifies the area of the coating for the object.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit

the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

See Also

Designed Equipment Properties Dialog Box (on page 304)

Select Insulation Material Dialog Box

Specifies the insulation materials for a selected object. By browsing through the part hierarchy, you can find and select an insulation material in the Catalog database.

 **Save**

This option is unavailable.

Cut

This option is unavailable.

 **Copy**

This option is unavailable.

 **Paste**

This option is unavailable.

 **Delete**

This option is unavailable.

Undo

This option is unavailable.

Insert Row

This option is unavailable.

Move Up

This option is unavailable.

Move Down

This option is unavailable.

Properties

Displays the properties of the selected object. Because you cannot modify any properties until the object is placed, all properties on the **Properties** dialog box are read-only.

Preview

Displays a bitmap symbol of the selected object. The image file must be assigned to the object in the catalog reference data.

Filter

This option is unavailable.

Sort

This option is unavailable.

Customize Current View

This option is unavailable.

List View

Sets the dialog box to display insulation thickness values in a list view.

Grid View

Sets the dialog box to display insulation thickness values in a spreadsheet-style grid view.

Back

Returns you to the previously selected insulation material folder. Use this command to navigate through the hierarchy to the specific insulation material you need.

Forward

Sends you to the last selected insulation material folder that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific insulation material you need.

Up One Level

Brings up the next highest level of the hierarchy. Use this command to navigate through the hierarchy to the specific material you need.

Check Data

This option is unavailable.

CheckData Ignored Inconsistencies

This option is unavailable.

Smart 3D Help

This option is unavailable.

Address

Sets the current location within the hierarchy of available insulation materials. The dropdown box lists the folders you have visited.

Definition Tab (Equipment Properties Dialog Box)

Displays the part information for the equipment object, the properties and their values, as defined in the reference data. If more than one equipment object is selected, only the common properties for the selected objects display on the tab. For more information about the information defined in the reference data, see the *Equipment and Furnishings Reference Data Guide,* available from the **Help > Printable Guides** command in the software.

Category

Select the properties to review. Equipment information is divided into several different categories, such as **Standard**, **Equipment Specification**, **Equipment Support**, **Nozzle Length**, and **Equipment Dimensions**.

IMPORTANT Not all categories are available for all equipment objects.

Property

Displays the name of the property. The properties that appear are dependent on the equipment type selected. For example, the properties displayed for an equipment object based on a pump are different from those based on a cooler. For more information on equipment part properties, see *the Equipment and Furnishings Reference Data Guide*, available with the **Help** > **Printable Guides** command within the Equipment and Furnishings task.

Value

Displays the value of the corresponding property.

Insulation Tab

Displays insulation properties that have been defined for the selected equipment.

Insulation Specification

Specifies whether or not the selected equipment object is insulated. The options on this dialog box remain disabled unless **User Defined** is selected from the dropdown list.

Purpose

Specifies the insulation purpose for the selected equipment object.

Material

Displays the **Select Insulation Material** dialog box from which you can specify an insulation

material. For more information, see *Select Insulation Material Dialog Box* (on page 294).

Thickness

Defines the thickness of the specified insulation material. This option remains disabled until a material is selected from the **Select Insulation Material** dialog box. By default, the smallest thickness value of the selected material is displayed. An alternative value may be selected from the dropdown list.

See Also

Designed Equipment Properties Dialog Box (on page 304)
Equipment Properties Dialog Box (on page 287)

SECTION 19

Place Designed Equipment Component

Specifies any equipment component from the Equipment Components folder of the catalog and places an occurrence of it inside the model. You can modify the offset of the component, its relationships to other equipment, and other properties during or after placement. Using positioning relationships, the **Place Designed Equipment Component** command allows you to mate, connect, or align equipment components, and you can use common tools like the **PinPoint** command for precise positioning of the component.

The goal of the software's designed equipment component modeling capabilities is to allow you to build an equipment component definition in the Model database by combining basic shapes, ports, such as nozzles, foundations, electrical, and so on, and properties defined in the reference data.

Designed equipment component objects can be built entirely of primitive geometric shapes. For example, you can use the **Place Designed Equipment Component** command to create a designed equipment component object in the model and add lugs, using the **Place Shape** command, to customize the designed equipment component.

You can also place a nozzle using a nozzle defined in the P&ID design basis (if P&ID design basis data is available).

Some equipment component shapes can be modeled more easily using solid modeling software. In addition to using the primitive shapes delivered with the software, you can import graphics from an SAT file as the geometry for the designed equipment shape. After the geometry is defined, you can then place ports to define distributed connections to the designed equipment.

Place Designed Equipment Component Ribbon

Sets options for adding designed equipment components to your model. This ribbon appears automatically after you select the **Place Designed Equipment Component** command and then select an equipment object, or when you select an existing equipment component.

TIP To find out the name of an option on the ribbon, pause the pointer over an option and read the ToolTip.

Equipment Component Properties

Edits the occurrence properties and reviews the static properties of an existing equipment component. Equipment components properties can be set and edited only after an equipment component object is placed in the model.

NOTE Any object modeled in Solid Edge that has occurrence properties cannot be placed or modified unless your computer has a copy of Solid Edge installed.

User Defined Form Definition

Displays **User Defined Form** for the particular equipment if it is defined in the catalog. See the *Equipment and Furnishings Reference Data Guide*, available from the **Help > Printable Guides** command in the software.

Relationship List

Lists all relationships for the selected equipment component and provides an option for creating a new relationship if the equipment component is not already fully constrained. An equipment component part is fully constrained when it has sufficient defined relationships to prevent movement or rotation of the part along all three coordinate axes.

Positioning Relationships

Displays the available options for types of positioning relationships. See Positioning Relationships for more information.

 **Delete Relationship**

Removes the selected relationship from the equipment component model and the database. Using the **Relationships** list box, select a previously existing relationship for the equipment component, and click **Delete Relationship**. You can use this command only when modifying existing equipment components.

 **Equipment Component Reference**

Prompts you for the reference on the equipment component to be placed that will be affected by the positioning relationship. In all cases, the part that you select in this step moves to create the relationship, and the part chosen in the **Second Part Reference** step remains fixed.

 **Second Part Reference**

Prompts you for the reference on the equipment component object or reference element already in the model that will be affected by the positioning relationship. After you select the reference, the software repositions the first equipment component part chosen with respect to the second part selected in the definition of the relationship.

Offset

Defines the offset distance for a mate or align relationship. Offsets are disabled when establishing a connect relationship. You can adjust this value after initial equipment component placement if needed.

Name

Displays the equipment component name, as dictated by your predefined name rules, and accepts changes to that name.

Equipment

Specifies the equipment with which to associate the selected equipment component.

Place designed equipment component

1. Click **Place Designed Equipment Component**  on the vertical toolbar.
2. Select the equipment to which to add the component from the model or the **Workspace Explorer**.

   *The **Select Equipment Component Type** dialog box appears.*
3. Select the equipment component to place, and click **OK**.

   *The **Design Equipment Component Properties** dialog box appears.*

4. Set properties as needed for the new designed equipment component, and click **OK**.

   The software returns you to the graphic window so you can define the position of the equipment component.

5. Using the **Place Designed Equipment Component** ribbon, do any of the following, if necessary:

   - Set an offset for the equipment relationship by typing the distance in the **Offset** box.
   - Click **Properties** on the ribbon, and type any necessary property information in the grid provided on the **Equipment Component Properties** dialog box.

     Edit Equipment Properties (on page 286)

   TIP You can view the definition properties of the equipment object using the **Properties** command within the **Select Equipment Component** dialog box on the equipment component property page after you place the equipment component. The occurrence properties for an equipment component object can be defined or modified after the equipment component object has been placed in the model.

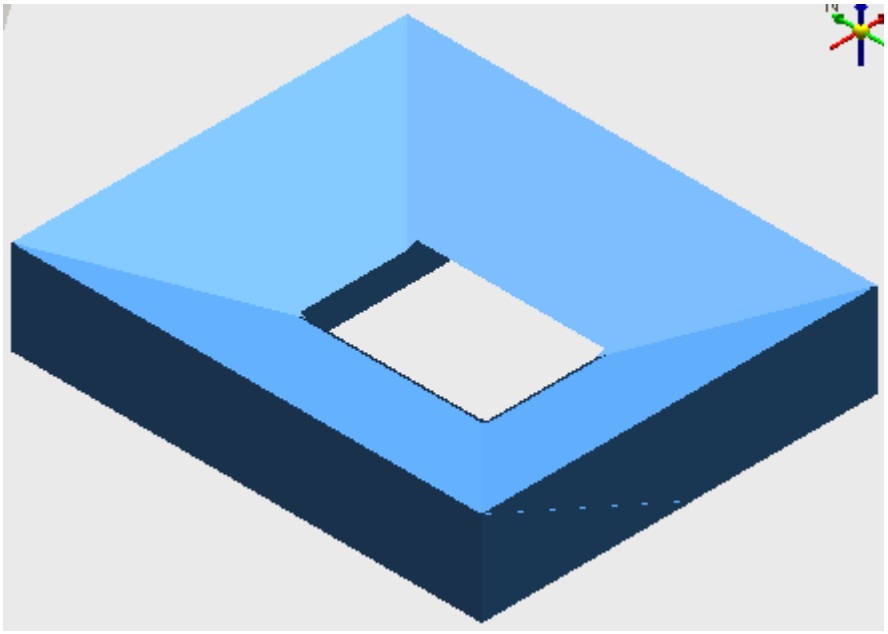
6. If necessary, add or change a positioning relationship by selecting it from the **Relationship** dropdown list on the **Place Equipment Component** ribbon.

   Set Positioning Relationships for Equipment (on page 283)

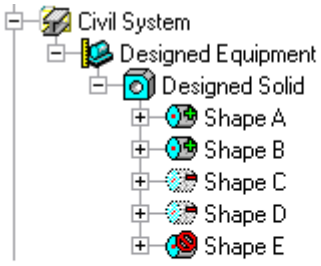
SECTION 20

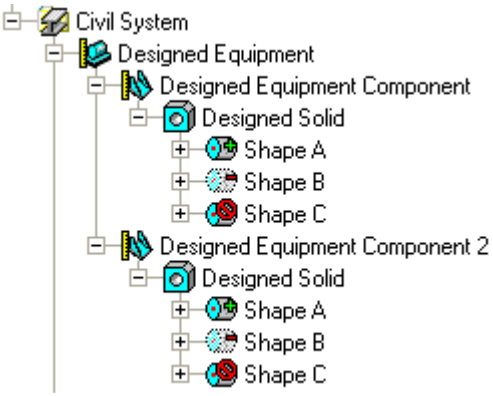
Place Designed Solid

The **Place Designed Solid** command is used to create highly customized designed solids for both complex concrete needs and equipment modeling.



Designed solids are essentially containers for a collection of shapes that you add to and subtract from the solid. To place a designed solid, a designed equipment or a designed equipment component must first be placed in the model. Then, the designed solid is placed as a child of the designed equipment or the designed equipment component. You can then add shapes underneath the designed solid. You can have multiple designed solids under one designed equipment or designed equipment component. You can also have multiple designed equipment components under one designed equipment.



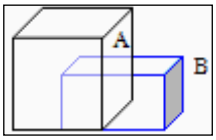


A shape must be a child of a designed solid in order for the software to compute the weight and center-of-gravity. The designed solid's weight and center-of-gravity properties are the sum of all shapes underneath a designed solid. For example, Shape A + Shape B + Shape C = Total Surface Area, Volume, Weight and Center-of-Gravity of the designed solid. If there is more than one solid underneath a designed equipment parent, these solids are also summed up to give a Total Weight and Center-of-Gravity for the designed equipment. For example, Solid A + Solid B + Solid C = Total Weight and Center-of-Gravity of the designed equipment.

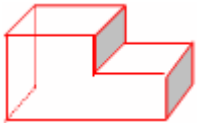
IMPORTANT Only shapes with simple physical or detailed physical aspects are included in weight and center-of-gravity calculations for designed solids. In addition, if some child shapes are displayed using the simple physical aspect and other child shapes of the same designed solid are displayed using the detailed physical aspect, only those shapes displayed as detailed physical are included in the weight and center-of-gravity calculations. If all the child shapes of a designed solid are displayed using the simple physical aspect, then they are all included in weight and center-of-gravity calculations. We recommend that you do not mix simple physical and detailed physical aspects in the same designed solid, as doing so will cause incorrect weight and center-of-gravity values.

Each shape that you add to the designed solid can add material to the parent solid, remove material from the parent solid, or not affect the parent (suppressed). The shape icon shown in Workspace Explorer represents an **Add**, **Subtract**, or **Suppress** operation. By default, shapes are added to a solid; however, during placement or after placement, you can change the shape operation to **Subtract Shape** or **Suppress Shape**.

Think of **Add Shape** and **Subtract Shape** as Boolean operations. For example, the designed solid below contains two shapes, A and B:



If you add shape B to shape A, the resulting designed solid appears as follows:



However, if you subtract shape B from shape A, the resulting designed solid appears as follows:



The Suppress Shape operation is used for creating construction geometries for reference when placing other shapes. The suppressed shape itself is not included in the designed solid; therefore, it will not affect weight and center-of-gravity calculations.

Place Designed Solid Ribbon

Sets options for adding designed solids to your model. This ribbon displays automatically after you select the **Place Designed Solids** command, or when you select an existing designed solid.

Designed Solid Properties

Edits the occurrence properties and reviews the static properties of an existing designed solid. Properties can be edited only after the object is placed in the model. For more information, see *Solid Properties Dialog Box* (on page 324).

Relationship List

Lists all relationships for the selected designed solid and provides an option for creating a new relationship if the designed solid is not already fully constrained. A designed solid part is fully constrained when it has sufficient defined relationships to prevent movement or rotation of the part along all three coordinate axes.

Positioning Relationships

Displays the available options for types of positioning relationships. Some options may not be available for all designed solid types. See Positioning Relationships for more information.

Delete Relationship

Removes the selected relationship from the designed solid in the model and in the database. Using the **Relationships** list box, select a previously-existing relationship for the designed solid, and click **Delete Relationship**. You can use this command only when modifying existing designed solids.

Shape Reference

Prompts you for the reference on the designed solid that will be affected by the positioning relationship. In all cases, the part that you select in this step moves to create the relationship, and the part chosen in the **Second Part Reference** step remains fixed.

Second Part Reference

Prompts you for the reference on the designed solid or reference element already in the model that will be affected by the positioning relationship. After you select the reference, the software repositions the first shape part chosen with respect to the second part selected in the definition of the relationship.

Offset

Defines the offset distance for a mate or align relationship. Offsets are disabled when establishing a connect relationship. You can adjust this value after initial designed solid placement, if needed.

Name

Displays the designed solid name, as dictated by your predefined name rules, and accepts changes to that name.

Equipment

Displays the name of the parent designed solid. This field only displays when you select a shape that comprises the designed solid.

Operators List

Activates the **Order Shapes** dialog box, with which you can control the order in which the software processes the shapes that comprise the designed solid. Controlling the order in which the shapes are processed can be important when some shapes that remove material from the designed solid overlap with other shapes that add material to the designed solid.

Order Shapes Dialog Box

Controls the order in which the designed solid's shapes are processed by the software, which can be very important when a shape that cuts material from the designed solid overlaps a shape that adds material to the designed solid. The designed solid could look very different depending on which shape, the cut or the add, the software processes last.

This dialog box is activated by **Operators List**  on the Modify Designed Solid ribbon.

Shape Name

Displays the name of the shape. You can select the shape name in the list to highlight that shape in the model view.

Operator Type

Displays whether the shape adds material to the designed solid or subtracts material from the designed solid.

Up

Moves the selected shape up one row in the list.

Down

Moves the selected shape down one row in the list.

Top

Moves the selected shape to be the first shape in the list.

Bottom

Moves the selected shape to the last shape in the list.

OK

Applies the changes made to the list to the designed solid and closes the dialog box.

Cancel

Ignores any changes made to the list and closes the dialog box.

Apply

Applies the changes made to the list to the designed solid, but leaves the dialog box active. Use this option to see changes that you make instantly.

What do you want to do?

- *Create a basic solid* (on page 321)
- *Import shape to designed solid* (on page 322)
- *Switch add, subtract, or suppressed shape* (on page 323)
- *Order shapes in designed solid* (on page 323)

Create a basic solid

1. Place a designed equipment or a designed equipment component using the **Equipment** > **Civil** > **Miscellaneous** equipment type.

   Place designed equipment (on page 301)

   Place designed equipment component (on page 315)

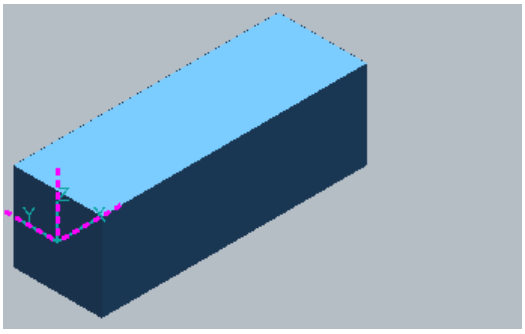
   TIP You must use the Equipment > Civil > Miscellaneous equipment type for the software to correctly calculate the mass properties of the solid (volume, surface area, weight, and center of gravity).

2. With the designed equipment or designed equipment component that you created in step 1 selected, click **Place Designed Solid** .

   The software adds the new designed solid as a child of the designed equipment or equipment component.

3. Click **Place Shape** , and then select the first shape for your designed solid.

4. In the **Shape Properties** dialog box, type the dimensions of your first shape, and then click **OK**.

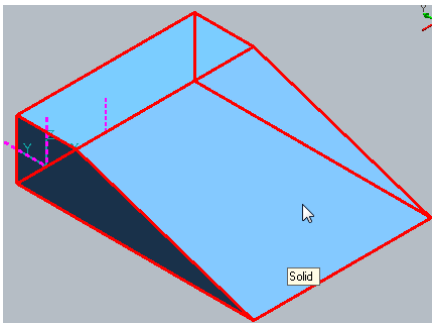
   The software adds the shape to the designed solid in Workspace Explorer and places the shape in the model at the point that you indicate.

5. On the ribbon, select whether the shape adds , subtracts , or is suppressed . Use suppressed for construction graphics and for shapes you want to temporarily remove from affecting the design solid.

6. Click **Place Shape** , and then select the next shape for your designed solid.

7. In the **Shape Properties** dialog box, type the dimensions for the second shape, and then click **OK**.

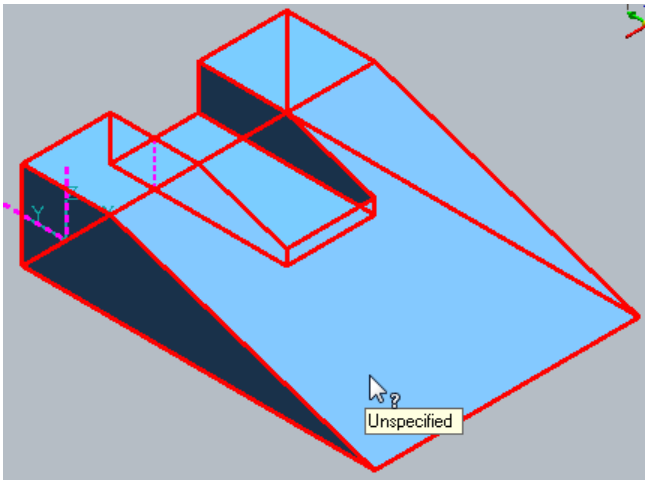
8. Identify the location of the second shape in the model.



TIP If you are placing a cylinder, cone, or eccentric cone shape, use the **Shape Reference** option on the toolbar to select the end plane of the cylinder or cone when using the Mate relationship to a plane of another shape.

9. On the ribbon, select whether the second shape adds , subtracts , or is suppressed .

10. Continue to added shapes to the designed solid using the add, subtract, and suppress options as needed.



Import shape to designed solid

1. Click **Place Imported Shape from File** on the vertical toolbar.
2. Select the parent designed solid in a graphic view or in the **Workspace Explorer**.
3. In the **Select Shape File** dialog box, navigate to and select the SAT file or MicroStation DGN file that contains the shape to insert. Click **OK**.
4. In the **Display Aspect** list box, select the aspects from the list.

   TIP You can click **Format** > **View** and change the **Render Selected Aspects** option in the **Format View** dialog box to display the aspect in the active graphic view.
5. Click **OK**.

6. Click in the graphic view to select an approximate location or reference element for a relationship for the shape.

SAT File Limitations

- From a performance point of view, the SAT file size should be less than 5 MB.
- From a drawings point of view, the files must not be detailed and must not spread across a large space as they are caught in volume filters for Drawings.
- Spreading the SAT files across multiple equipment will reduce the volume of each .sat file.
- For efficient Interference Checker processing, SAT files should not have a footprint (range) greater than 100 meters.

Switch add, subtract, or suppressed shape

1. Click **Select** on the vertical toolbar.
2. Set the locate filter to **Shape**.
3. In a graphic view or in Workspace Explorer, select the shape to edit.
4. On the ribbon, select whether the shape adds, subtracts, or is suppressed.

Order shapes in designed solid

1. Click **Select** on the vertical toolbar.
2. Set the locate filter to **Solids**.
3. Select the parent designed solid whose shapes you want to re-order.
4. Click **Operators List** on the ribbon.
5. In the **Order Shapes** dialog box, use the **Up**, **Down**, **Top**, and **Bottom** commands to order the shapes as needed.
6. Click **Apply** or **OK** to save your changes and update the designed solid.

TIP See *Place Designed Solid* (on page 317) for more information on why the order of shapes in a solid is important.

Solid Properties Dialog Box

Displays designed solid properties for review and editing.

See Also

Occurrence Tab (Solid Properties Dialog Box) (on page 324)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

Occurrence Tab (Solid Properties Dialog Box)

Displays all editable instance-specific information about the selected designed solid. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If more than one solid object is selected, common occurrence properties for the selected objects appear on the tab.

When viewing properties for a single solid object, the following properties appear. More properties may appear depending on what you defined in the reference data. For more information about occurrences defined in the reference data, see the *Equipment and Furnishings Reference Data Guide* available from the **Help** > **Printable Guides** command in the software.

NOTE Because equipment properties are customizable in the equipment and furnishings reference data, only the properties that are required by the software are documented.

Category

Select the properties to view, modify, or define. Equipment properties are divided into several different categories: Standard, Weight and CG, and Position and Orientation.

Display Aspect

Select the display aspects that you want to see for the solid.

Standard

Name

Displays the name of the solid object. The solid name is based on the **Name Rule** selection. If you type a name in this field, the **Name Rule** property updates to **User Defined**.

Name Rule

Specify the naming rule to use to name this solid object. You can select one of the listed rules or select **User Defined** to specify the solid name yourself in the **Name** box.

Material Name

Select the material for the solid.

Material Grade

Select the material grade for the solid.

NOTE You must define a material and material grade before the software can calculate the solid's weight and center of gravity.

Surface Area

Displays the calculated surface area.

Volume

Displays the calculated volume.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material density multiplied by the object's solid volume. Therefore, the material of the object affects the weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

NOTE For equipment, the **Weight and CG** property **Wet Weight** is the sum of **Dry Weight** and **Water Weight**. The dry weight and water weight values are catalog properties entered on the part sheet for the equipment.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Wet CG X

Specifies the X-axis location of the wet center-of-gravity.

Wet CG Y

Specifies the Y-axis location of the wet center-of-gravity.

Wet CG Z

Specifies the Z-axis location of the wet center-of-gravity.

Dry WCG Origin

Specifies the way in which the dry weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the dry weight center-of-gravity location relative to the active coordinate system.

Wet WCG Origin

Specifies the way in which the wet weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the wet weight center-of-gravity location relative to the active coordinate system.

Position and Orientation

East

Displays the distance of the connection point from the active coordinate system origin in the east direction.

North

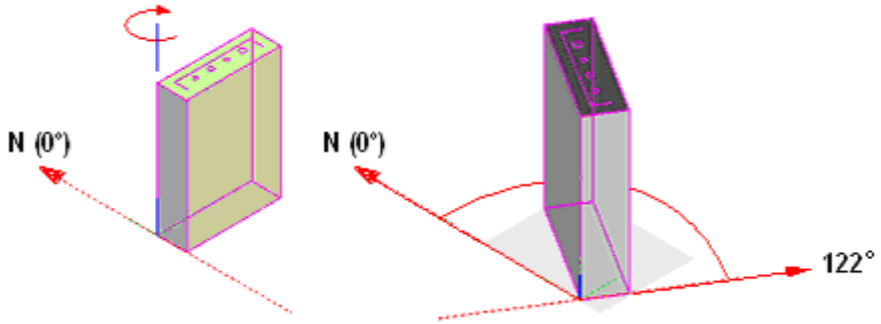
Displays the distance of the connection point from the active coordinate system origin in the north direction.

Elevation

Displays the distance of the connection point above or below the active coordinate system origin.

Bearing

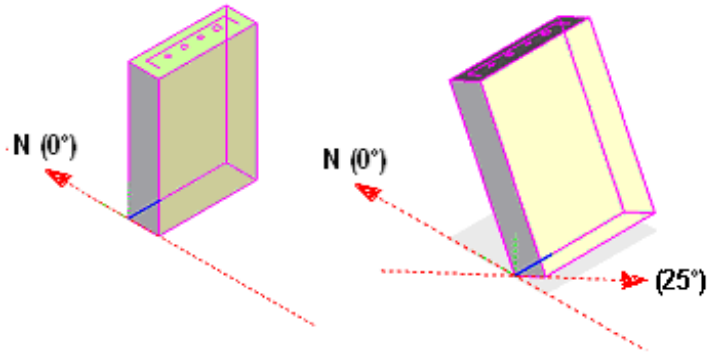
Specifies the bearing angle for the object. The **Bearing** angle is measured between the local x-axis of the object and the Y-axis (North) of the global coordinate system in the XY-plane. The local x-axis is the default axis of primary symmetry for all symbols in the catalog. The bearing measurement direction is clockwise from the active coordinate system North looking in the negative active coordinate system direction; that is, down from 0 to 360 degrees. You can enter negative bearing angles, but the software automatically converts them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the bearing measure displays 0 degrees.



Pitch

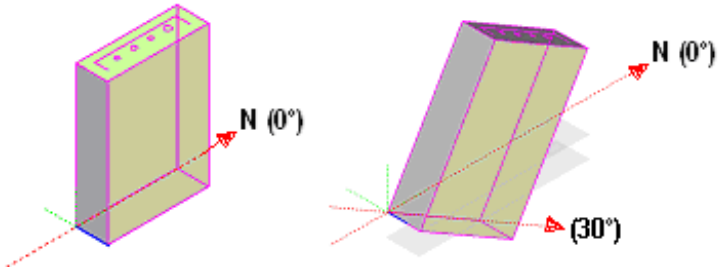
Specifies the pitch angle for the shape. The **Pitch** angle measures between the x-axis of the object and the X-axis of the global coordinate system in the XZ-plane. This option sets the reference in the model to a line that is the intersection of the vertical plane through the X-axis of the local coordinate system and the active coordinate system horizontal plane. The angle is measured in the positive direction from the horizontal plane in the active coordinate system up direction regardless of the current bearing. Another way to describe pitch is the rotation of the object about its y-axis. Pitch angles are limited to between -90 degrees and

+90 degrees, with 0 indicating horizontal.



Roll

Specifies the roll angle for the object. The **Roll** angle measures between the local z-axis of the object and the Z-axis of the global coordinate in the YZ-plane. Another way to describe roll is the rotation of the object about its x-axis. This option sets the reference in the model to a line that is perpendicular to the local coordinate system x-axis and in the horizontal plane. Roll angles are measured clockwise from horizontal to the y-axis of the local coordinate system. The roll angle is between 0 and 360 degrees. You can enter negative roll angles, but the software automatically converts them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the reference in the model is the North axis.



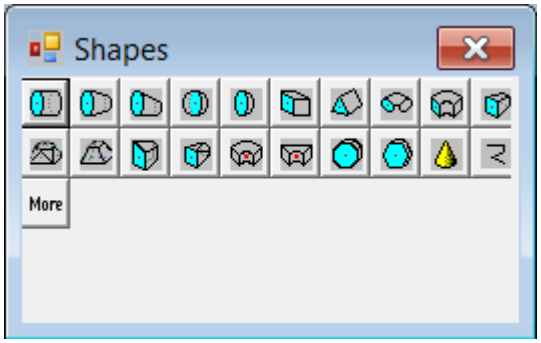
SECTION 21

Place Shape

Adds additional shapes to an existing equipment or equipment component type. The **Place Shape** command uses the equipment or equipment component object selected in the **Workspace Explorer** hierarchy. If no equipment or equipment component object is selected prior to starting the command, you are prompted to select an equipment or equipment component object.

If you click and hold the button on the vertical toolbar, the **Place Shape** palette displays so that you can select a different type of shape to place.

Click **More** to display the **Select Shape** dialog box and select a shape from those available in the Catalog database.



Place Shape Ribbon

Sets options for positioning the specific shape as part of a selected designed equipment object.

Properties

Edits the occurrence properties and reviews the static properties of an existing shape. Properties can be edited only after the object is placed in the model. For more information, see *Shape Properties Dialog Box* (on page 333).

Relationship List

Shows the relationship currently applied to the shape and allows you to change or add new relationship as needed.

Positioning Relationship

Displays the available options for types of positioning relationships: Mate, Align, and Connect. Some options may not be available for all shape types. For more information, see Positioning Relationships.

Delete Relationship

Removes the selected relationship from the model. Using the **Relationships List** box, select a previously existing relationship for the shape, and click **Delete Relationship**. You can use this command only when modifying existing shapes. This control is available only after a shape has been added to the equipment or equipment component.

Shape Reference

Prompts you for the reference on the shape to be placed that will be affected by the positioning relationship. In all cases, the part that you select in this step moves to create the relationship, and the part chosen in the **Second Part Reference** step remains fixed.

Second Part Reference

Prompts you for the reference on the shape object already in the model that will be affected

by the positioning relationship. After you select the reference, the software repositions the first shape chosen with respect to the second part selected in the definition of the relationship.

Offset

Defines the offset distance for a mate or align relationship. Offsets are disabled when establishing a connect relationship. You can adjust this value after initial equipment placement if needed. This control is only available after a shape has been added to the designed equipment.

Name

Displays the shape name, as dictated by your predefined name rules, and accepts changes to that name.

NOTES

- If the name of the parent equipment changes, the software automatically updates the name of the shape to reflect that change. For example, if the name of the parent item changes from Pump01 to Pump100, the shape's name changes from Pump01-Shape-001 to Pump100-Shape-001.
- Some properties for the shape may be read-only on the **Place Shape** ribbon, depending on the type of shape you selected.

Equipment

Specifies the equipment or equipment component object with which to associate the selected shape.

Add Shape/Subtract Shape/Suppress Shape

Defines how the shape is used for a designed solid. This option is only available when you are placing or editing a shape in a parent designed solid.

 **Add Shape**

Select to have the shape add its material to the parent designed solid.

 **Subtract Shape**

Select to have the shape subtract its material from the parent designed solid.

 **Suppress Shape**

Select to have the shape added to the parent designed solid in the hierarchy, but have the shape's material ignored--neither added to nor subtracted from the parent designed solid. Use this option to temporarily remove a shape's effect on the parent designed solid, but not remove the shape from the designed solid entirely. You can use suppressed shapes as construction graphics for the placement of other shapes in the designed solid.

Select Shape Dialog Box

Specifies a shape to place on an equipment or equipment object. The dialog box appears automatically when you click **More** in the **Shape** floating palette. By browsing through the shapes hierarchy, you can find any shape that exists in the Catalog database. After you select a shape, the **Shape Properties** dialog box appears so you can define shape properties. When you click **OK** on the **Shape Properties** dialog box, the software returns you to the graphic window so you can place the shape.

Back

Returns you to the previously selected shape type or node. Use this command to navigate through the hierarchy to the specific shape you need.

Forward

Sends you to the last selected shape type or node that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific shape you need.

Up One Level

Brings up the next highest level of the Shapes catalog hierarchy. Use this command to navigate through the hierarchy to the specific shape you need.

Properties

Displays the properties of the selected shape. Because you cannot modify any properties until the shape is placed, all properties on the **Properties** dialog box are read-only.

Preview

Displays a bitmap symbol of the selected shape. The image file must be assigned to the shape in the catalog reference data.

List View

Sets the dialog box to display shapes in a list view.

Grid View

Sets the dialog box to display shapes in a spreadsheet-style grid view.

Address

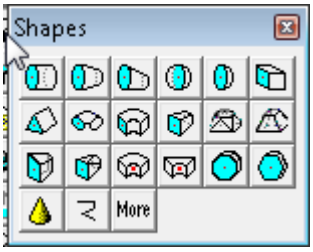
Specifies your exact location within the displayed hierarchy.

What do you want to do?

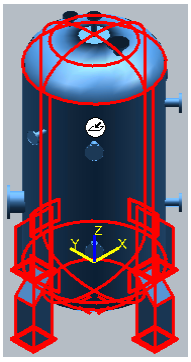
- *Place a shape* (on page 331)
- *Edit shape properties* (on page 332)
- *Edit prismatic shape properties* (on page 333)

Place a shape

1. Click **Place Shape** on the vertical toolbar, and hold down a few seconds to display the floating **Shapes** palette.

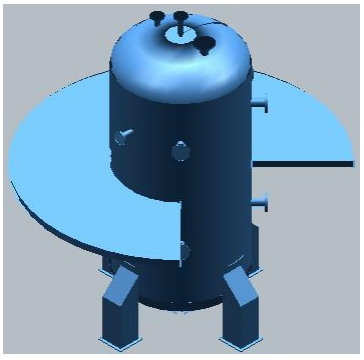


2. Select the shape to place from the **Shapes** palette. The icon of the last used shape displays on the toolbar.



TIP If an equipment or an equipment component object has not been selected, you are prompted to select one. Do this either in a graphic view or in the system hierarchy in the **Workspace Explorer**.

3. If necessary, make adjustments on the **Shape Properties** dialog box, and click **OK**.
4. Click in the graphic view to select an approximate location or reference element for a relationship for the shape.



NOTES

- You can continue using the **Place Shape** command to combine geometric shapes and create customized equipment or equipment component objects.
- After a shape has been placed in the model as part of the equipment or equipment component object, you can use the horizontal ribbon to add or change the positioning relationship.
- Press the left or right arrow keys to rotate the shape by 90-degree increments at any time during the placement of the shape. Press the up arrow to scroll through the three possible axes of rotation. Press the down arrow key to scroll to each datum point.

Edit shape properties

1. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a shape in the graphic window.

   TIP If the **Locate Filter** box is not displayed on the horizontal ribbon, click **Select Tool** on the vertical toolbar.

2. Click **Shape Properties** on the horizontal ribbon.

   TIP You can also access the **Shape Properties** dialog box by selecting **Properties** from the **Edit** menu, or by right-clicking the shape and choosing **Properties** from the short-cut menu.

3. On the *Occurrence Tab (Shape Properties Dialog Box)* (on page 334), modify the geometric dimensions of the shape.
4. Access the other dialog box tabs, and modify the properties as needed.
5. Click **OK** to save your changes and return to the workspace.

   TIP You can also select **Apply** to put the changes into effect and continue working in the **Properties** dialog box.

NOTES

- Each geometric shape object has its own **Properties** dialog box that displays its corresponding parameters, including any reference graphics that illustrate what the dimensional parameters represent.
- All occurrence and definition properties for an equipment object are defined by the Excel workbook named **Equipment.xls** in the reference data. For information on adding or editing different types of properties, refer to the *Reference Data Guide,* accessible from the **Help > Printable Guides** command.

Edit prismatic shape properties

1. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.

   TIP If the **Locate Filter** box is not displayed on the horizontal ribbon, click the **Select Tool** on the vertical toolbar.

2. Click **Properties** on the horizontal ribbon.

   TIP You can also access the **Prismatic Shape Properties** dialog box from the **Relationship** tab on the **Designed Equipment Properties** dialog box. Select the name of the prismatic shape to edit, and click **Go To**.

3. On the **Occurrence** tab, modify the display aspect or the name of the prismatic shape.
4. On the **Cross-Section** tab, modify the geometric dimensions of the existing cross-section, or select a new cross-section and type new property information.
5. Access the other dialog box tabs, and modify the properties as needed.
6. Click **OK** to save your changes and return to the workspace.

   TIP You can also select **Apply** to put the changes into effect and continue working in the **Properties** dialog box.

NOTES

- Each geometric shape object has its own **Properties** dialog box that displays its corresponding parameters, including any reference graphics that illustrate what the dimensional parameters represent.
- All occurrence and definition properties for an equipment object are defined by the Excel workbook named **Equipment.xls** in the reference data. For information on adding or editing different types of properties, refer to the *Reference Data Guide,* accessible from the **Help > Printable Guides** command in the software.

Shape Properties Dialog Box

Displays shape properties for review and editing.

See Also

Occurrence Tab (Shape Properties Dialog Box) (on page 334)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

Occurrence Tab (Shape Properties Dialog Box)

Sets properties for the unique instance of the selected shape. Shapes can be placed either from what has been defined in the reference data, or by importing a geometric shape contained within an SAT file.

Category

Displays the defined category name for the selected designed equipment. This data is retrieved from the Equipment and Furnishings reference data.

Display Aspect

Displays the **Display Aspect** dialog box which lists all the display aspects that may be applied to the selected shape.

Property

Lists all the dimensional properties of the selected shape. You can add or modify these properties through the catalog reference data. The properties available change depending on the type of shape selected.

Value

Shows the current values for all properties of the selected shape. You can modify these values to reflect exact design needs or deviations from the standard part.

Name

Specifies the name of the object. If a **Name Rule** is specified, then the software uses that rule to determine this name. If the **Name Rule** value is **User Defined**, then you must type a name in this box.

Name Rule

Displays the available name rules for the selected object. Specify the naming rule to use to name the object. You can select one of the listed rules, or you can select **User Defined** to specify the name yourself in the **(Name)** box. **Behavior Controlled by User**

Indicates whether or not you can delete the object. Select **False** to prevent anyone from deleting the object from the model.

Prismatic Shape Properties Dialog Box

Displays prismatic shape properties for review and editing.

See Also

Occurrence Tab (Prismatic Shape Properties Dialog Box) (on page 335)
Cross-Section Tab (Prismatic Shape Properties Dialog Box) (on page 335)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

Occurrence Tab (Prismatic Shape Properties Dialog Box)

Sets properties for the unique instance of the selected prismatic shape. Prismatic shapes can be placed in the model by selecting **PrismaticShape** in the **Shapes** dropdown list on the *Occurrence Tab (Shape Properties Dialog Box)* (on page 334).

Category

Displays the defined category name for the selected prismatic shape. This data is retrieved from the Equipment and Furnishings reference data.

Shape

Displays the **PrismaticShape** icon. This option is read-only when you access the tab to edit a prismatic shape that has already been placed in the model.

Display Aspect

Lists all the display aspects that can be applied to the selected shape.

Name

Displays the name defined for the selected prismatic shape during its initial placement in the model.

Name Rule

Displays the available name rules for the selected object. Specify the naming rule to use to name the object. You can select one of the listed rules, or you can select **User Defined** to specify the name yourself in the **(Name)** box.

Cross-Section Tab (Prismatic Shape Properties Dialog Box)

Displays information about the cross-section for a prismatic shape that was projected along a path.

Cross Section

Specifies the type of cross-section for the object. You can either select a standard cross-section defined in the reference data or select **Sketch** to draw your own cross-section.

Select any standard cross-section type in the **Cross Section** list to modify the properties that are described in this topic. When you select **Sketch** in the **Cross Section** list, you can view options, such as each point, its X-, Y-, and Z-coordinates in the model, and its turn type, but you cannot modify them.

For more information, see *General Tab (Sketch Properties Dialog Box)* (on page 362).

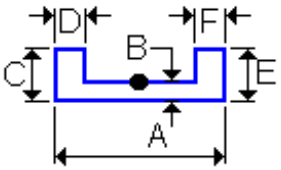
Display Cross Section Image

Displays the image associated with the standard cross-section type in the reference data.

Cross-section images show the dimensions that you can define for the cross-section type, the default cardinal points for the cross-section type, and the angle for the cross-section. For example, the following graphic contains the dimensions and cardinal point for a standard road cross-section:

If no image is associated with the cross-section and the selected cardinal point, **Image Not Available** displays at the bottom of the tab.

A - G



Defines the dimensions for standard cross-sections. If an image is defined for the cross-section in the reference data, you can see what each letter represents by clicking **Display Cross Section Image** .

Cardinality

Defines the point where the software attaches the cross-section to the path. If you click **Display Cross-Section Image** to see a graphic of the cross-section, you can view where each cardinal point is located by selecting each cardinal point in the list. The software automatically updates the display, if the appropriate graphic is available, with a graphic containing the selected cardinal point.

You can select **User Defined** cardinal points to specify a reference point on selected cross section for precise placement.

Angle

Defines the angle by which the cross-section is rotated about the path.

SECTION 22

Creating Customized Shapes

In the Equipment and Furnishings task, you can create customized shapes in the location that you need using the place prismatic shape feature. This feature is especially useful for designed equipment objects that need to have an unusual shape.

When you place a prismatic shape, a two-dimensional cross-section is projected along a path that you specify in the model to create the shape. The path determines the actual location of the shape in the model. The cross-section, on the other hand, defines the shape and its dimensions.

Defining Paths

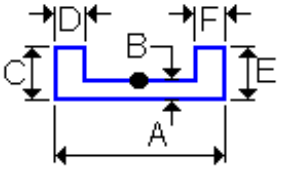
When you define the path along which the cross-section will be projected, you can choose from straight lines or arcs. You can also control all aspects of the path by specifying the types of turns that you need, the dimensions of the turns, and the plane for the path.

Defining Cross-sections

When you define the cross-section for the shape, you can select from a standard set of cross-sections, defined in the reference data, or you can sketch your own two-dimensional cross-section.

When you use a standard cross-sectional type, you can change the dimensions and the cardinal point of the cross-section on the **Cross-Section** tab of the **Shape Properties** or **Prismatic Shape Properties** dialog box. The cardinal point of the cross-section is important in determining the shape and location of the resulting volume. The cardinal point is the point where the software attaches the cross-section to the path. All standard cross-section types have various cardinal points from which you can choose.

For example, in the following graphic, you can see that the cardinal point is located in the center of the road cross-section. If the cardinal point were moved to another location, the actual path of the roadway would be different.



When you sketch a cross-section, you must sketch the cross-section on the two-dimensional plane that is orthogonal to the first leg of the path. The software displays this plane, which is perpendicular to the path, as you sketch the cross-section. The cardinal point is defined as you sketch. In other words, the cross-section surrounds the path and is attached to the path exactly as you sketch it.

Place Prismatic Shape Ribbon

Sets options for placing shapes defined by a path and the cross-section projected along the path.

Shape Properties

Displays the **Prismatic Shape Properties** dialog box, on which you can set properties for the shape that you need to place. For more information, see Prismatic Shape Properties Dialog Box.

Path

Displays the **Create Path** ribbon, which defines the path along which the cross-section is projected.

Cross-Section

Sets properties for the cross-section. If one of the standard cross-section types is selected in the **Cross-Section Type** box, this button is unavailable. If you select **Sketch** as the cross-section type, this button displays the **Create Path** ribbon to allow you to sketch the two-dimensional cross-section. The **Cross-Section** button is only available after you define a path for the volume or when you select **User Defined** for **Cardinality**.

Finish

Places the shape along the path specified with the specified cross-section.

Cross-Section Type

Specifies the type of cross-section to project along the specified path. You can either select a standard cross-section defined in the reference data, or you can sketch your own custom cross-section.

Relationship List

Lists all relationships for the selected shape and provides an option for creating a new relationship if the object is not already fully constrained. An object is fully constrained when it has sufficiently defined relationships to prevent movement or rotation of the object along all three coordinate axes.

Positioning Relationships

Displays the available options for types of positioning relationships: Mate, Align, and Connect. Some options may not be available for all shape types. See Positioning Relationships for more information.

Delete Relationship

Removes the selected relationship from the model. Using the **Relationships List** box, select a previously existing relationship for the shape, and click **Delete Relationship**. You can use this command only when modifying existing designed equipment. This control is available only after a shape has been added to the designed equipment.

Shape Reference

Prompts you for the face of the shape to be placed that will be affected by the positioning relationship. In all cases, the part that you select in this step moves to create the relationship, and the part chosen in the **Second Part Reference** step remains fixed. This control is available only after a shape has been added to the designed equipment.

Second Part Reference

Prompts you for the reference on the shape object already in the model that will be affected by the positioning relationship. After you select the reference, the software repositions the first shape part chosen with respect to the second part selected in the definition of the relationship.

Offset

Defines the offset distance for a mate or align relationship. Offsets are disabled when establishing a connect relationship. You can adjust this value after initial placement if needed. This control is available only after a shape has been added to the designed

equipment.

Name

Displays the shape name, as dictated by your predefined name rules, and accepts changes to that name.

Equipment

Specifies the designed equipment object with which to associate the selected shape.

Create Path Ribbon

Sets options for defining a new path.

 **Sketch Properties**

Displays the **Sketch Properties** *dialog box* (on page 362), in which you can view properties for the path.

Finish Path

Displays the path in the active view and returns to the model with the place prismatic shape feature still enabled.

Cancel

Cancels the changes you made and returns you to the model.

Edit

Modifies and moves the existing path. When you initially create a path, this option is available only after you place at least two points in the path. You can select the segment, turn, or multiple segments to which to make modifications.

Create

Sketches the path or adds segments to an existing path.

Origin

Defines the origin on the cross-section for prismatic shape construction.

 **Reference Point**

Specifies that you are currently defining the first point of the path segment.

 **End Point** (Straight Line)

Specifies that you are currently defining the second point of a straight path segment.

 **End Point** (Arc)

Specifies that you are currently defining the second point of an arc. This option appears only when **Arc** is selected in the **Line Type** list.

 **Third Point** (Arc)

Specifies that you are defining the final point of an arc.

Path Type

Specifies the type of line for the current segment in the path. To change the segment type, click a new type in the **Line Type** list.

Line Type Options

Line

Defines the line type for the segment to be a straight line.

Arc by 3 Points

Defines the line type for the segment to be an arc. To define the arc, you must click three points in the view.

No Line

Specifies that you do not want the current segment of the path to have a line associated with it.

Plane

Activates options for selecting a working plane for the path.

Working Plane Options

Plan Plane

Defines the work surface as the XY plane.

Elevation Plane: East-West

Defines the work surface as the XZ plane.

Section Plane: North-South

Defines the work surface as the YZ plane.

Plane by Turn

Defines the work surface as the plane defined by an existing turn. You select the turn to set the plane.

Plane by Three Points

Defines the work surface using three points that you define.

No Plane

Clears any work surfaces. The software does not project points that you place to any plane.

Lock Angle

Locks or unlocks the **Angle** box. Locking the corresponding angle value creates a constraint along which the selected turn angle can be moved.

Angle

Specifies the angle for the turn.

Lock Length

Locks or unlocks the **Length** box.

Length

Specifies the length of the selected path segment.

Turn Type

Specifies the type of turn associated with the current path segment. The **Turn Type** option is unavailable if you select **Arc** or **No Line** in the **Line Type** list.

Turn Type Options

None

Indicates that no special turn type will be applied to the turn.

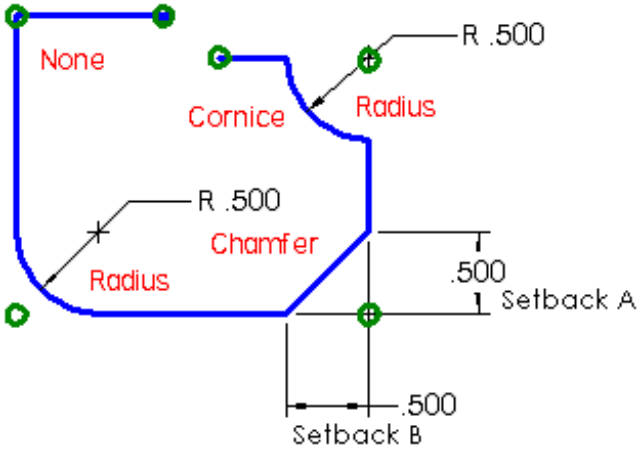
Bend

Specifies that the turn type between two segments is a bend. You can specify the angle of the bend in the **Feature Value** box.

Cornice

Specifies that the turn type between two segments is a cornice. You can specify the radius of the cornice in the **Feature Value** box.

The following graphic includes an example of each of the available turn types:



Turn Type Value

Defines dimensions for the selected turn type.

Edit Path Straight Segment Ribbon

Sets options for modifying a straight segment of an existing path.

Sketch Properties

Displays the *Sketch Properties dialog box* (on page 362), on which you can view properties for the path.

Finish Path

Displays the path in the active view and returns to the model with the place prismatic shape feature still enabled.

Cancel

Cancels the changes you made and returns you to the model.

Edit

Modifies and moves the existing path. You can select the segment, turn, or multiple segments to which to make modifications.

Create

Displays the **Create Path** ribbon to allow you to add segments to an existing path.

Origin

Defines the origin on the cross-section for prismatic shape construction.

 **Reference Point**

Specifies that you are selecting the starting location of the move vector.

 **End Point**

Specifies that you are selecting the ending location of the move vector.

Path Type

Specifies the type of line for the current segment in the path. To change the segment type, click a new type in the **Line Type** list.

Path Type Options

 **Line**

Defines the line type for the segment to be a straight line.

 **Arc by 3 Points**

Defines the line type for the segment to be an arc. To define the arc, you must click three points in the view.

 **No Line**

Specifies that you do not want the current segment of the path to have a line associated with it.

Plane

Activates options for selecting a working plane for the path.

Working Plane Options

 **Plan Plane**

Defines the work surface as the XY plane.

 **Elevation Plane: East-West**

Defines the work surface as the XZ plane.

 **Section Plane: North-South**

Defines the work surface as the YZ plane.

 **Plane by Turn**

Defines the work surface as the plane defined by an existing turn. You select the turn to set the plane.

Plane by Three Points

Defines the work surface using three points that you define.

No Plane

Clears any work surfaces. The software does not project points that you place to any plane.

Delete Selected Items

Deletes the selected path segments.

Length Locked

Defines whether or not the length of the selected segment should remain constant while moving.

When locked , the software automatically modifies the turn points, along with the length and angle of adjacent segment, to remain connected to the moved segment. The length of the moved segment does not change.

When not locked , the software extends or shortens the associated segments to connect with the new position of the moved segment. The length of the moved segment can change.

Edit Path Arc Ribbon

Sets options for modifying an arc that is part of an existing path.

Sketch Properties

Displays the *Sketch Properties dialog box* (on page 362), on which you can view properties for the path.

Finish Path

Displays the path in the active view and returns to the model with the place prismatic shape feature still enabled.

Cancel

Cancels the changes you made and returns you to the model.

Edit

Modifies and moves the existing path. You can select the segment, turn, or multiple segments to which to make modifications.

Create

Displays the **Create Path** ribbon to allow you to add segments to an existing path.

Origin

Defines the origin on the cross-section for prismatic shape construction.

Reference Point

Specifies that you are selecting the starting location of the move vector.

End Point

Specifies that you are selecting the ending location of the move vector.

Path Type

Specifies the type of line for the current segment in the path. To change the segment type, click a new type in the **Line Type** list.

Path Type Options

Line

Defines the line type for the segment to be a straight line.

Arc by 3 Points

Defines the line type for the segment to be an arc. To define the arc, you must click three points in the view.

No Line

Specifies that you do not want the current segment of the path to have a line associated with it.

Working Plane Options

Plan Plane

Defines the work surface as the XY plane.

Elevation Plane: East-West

Defines the work surface as the XZ plane.

Section Plane: North-South

Defines the work surface as the YZ plane.

Plane by Turn

Defines the work surface as the plane defined by an existing turn. You select the turn to set the plane.

Plane by Three Points

Defines the work surface using three points that you define.

No Plane

Clears any work surfaces. The software does not project points that you place to any plane.

Delete Selected Items

Deletes the selected path segments.

Edit Path Turn Ribbon

Sets options for modifying a turn in an existing path.

Sketch Properties

Displays the *Sketch Properties dialog box* (on page 362), on which you can view properties for the path.

Finish Path

Displays the path in the active view and returns to the model with the place prismatic shape

feature still enabled.

Cancel

Cancels the changes you made and returns you to the model.

Edit

Modifies and moves the existing path. You can select the segment, turn, or multiple segments to which to make modifications.

Create

Displays the Create Path Ribbon to allow you to add segments to an existing path.

Origin

Defines the origin on the cross-section for prismatic shape construction.

 **Reference Point**

Specifies that you are selecting the starting location of the move vector.

 **End Point**

Specifies that you are selecting the ending location of the move vector.

Plane

Activates options for selecting a working plane for the path.

Working Plane Options

 **Plan Plane**

Defines the work surface as the XY plane.

 **Elevation Plane: East-West**

Defines the work surface as the XZ plane.

 **Section Plane: North-South**

Defines the work surface as the YZ plane.

 **Plane by Turn**

Defines the work surface as the plane defined by an existing turn. You select the turn to set the plane.

 **Plane by Three Points**

Defines the work surface using three points that you define.

 **No Plane**

Clears any work surfaces. The software does not project points that you place to any plane.

 **Delete Selected Items**

Deletes the selected path segments.

Angle 1

Displays the first angle used in the turn, if one exists. This option is read-only.

Angle 2

Displays the second angle used in the turn, if one exists. This option is read-only.

Angle 3

Displays the third angle used in the turn, if one exists. This option is read-only.

Turn Type

Specifies the type of turn. You can change the turn type by selecting another type in the list.

Turn Type Options

None

Indicates that no special turn type will be applied to the turn.

Bend

Specifies that the turn type between two segments is a bend. You can specify the angle of the bend in the **Feature Value** box.

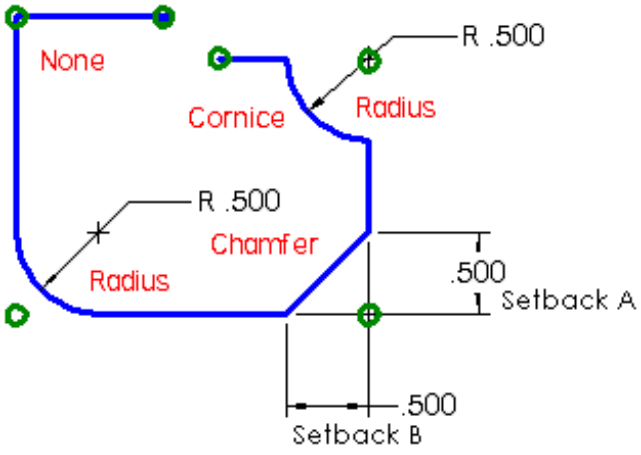
Cornice

Specifies that the turn type between two segments is a cornice. You can specify the radius of the cornice in the **Feature Value** box.

Chamfer

Specifies that the turn type between two segments is a chamfer. You can specify the dimensions for setback A and setback B of the chamfer in the **Feature Value** box. The dimensions of setback A and setback B for the chamfer must be the same.

The following graphic includes an example of each of the available turn types:



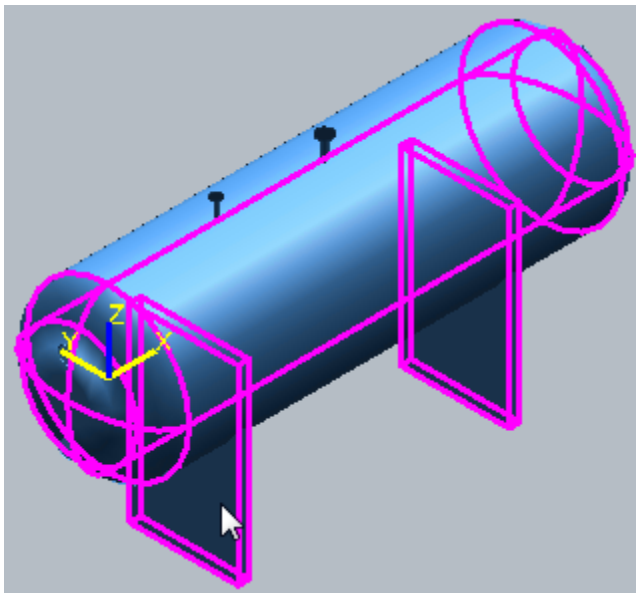
Turn Type Value

Defines dimensions for the selected turn type.

Place a prismatic shape

1. Click **Place Shape** on the vertical toolbar.

   TIP If a designed equipment object has not been selected, select one in a graphic view or in the **Workspace Explorer**.

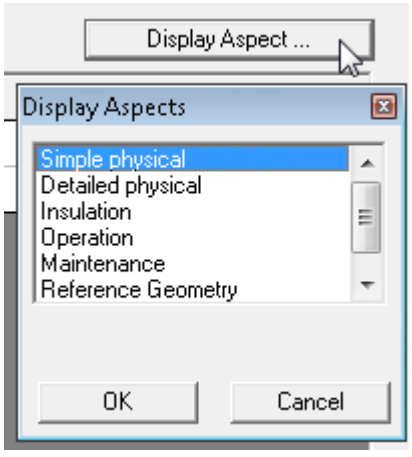


   *The **Shapes** palette displays*

2. On the **Shapes** dialog box, select **Prismatic Shape**.

   TIP Click **More** to select a shape from the catalog browser.

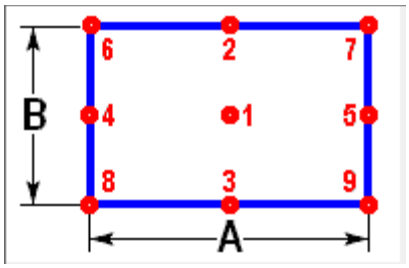
3. Click **Display Aspect** and select the aspects from the list.



   TIP You can click **Format** > **View** and change the **Render Selected Aspects** option in the **Format View** dialog box to display the aspect in the active graphic view.

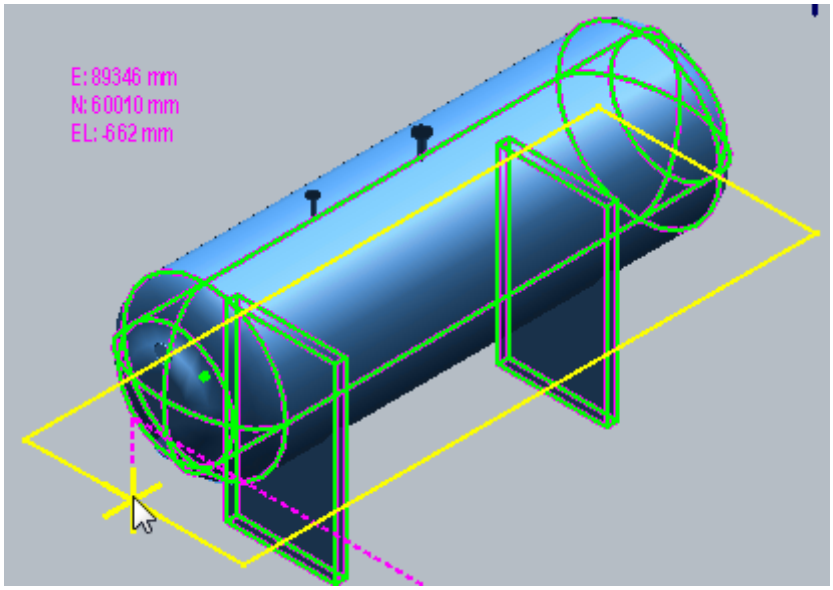
4. Specify a name for the new shape in the **Name** box.

5. On the **Cross-Section** tab, select a cross-section in the drop-down list.



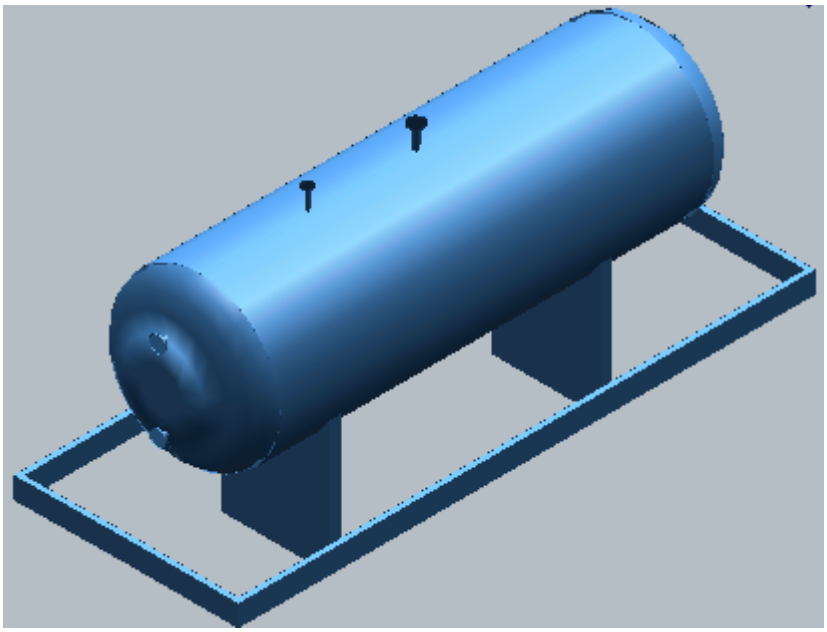
A corresponding image of the selected cross-section appears at the bottom of the dialog box.

6. In the **Value** column, type the appropriate geometric dimensions, cardinal point, and angle.
7. Click **OK** to return to the model.
8. Define the path for the shape. For more information, see *Define the Path for a Prismatic Shape* (on page 349).



9. Click **Finish Path** to finish the path.

10. Click **Finish** to place the new shape and save it to the database.



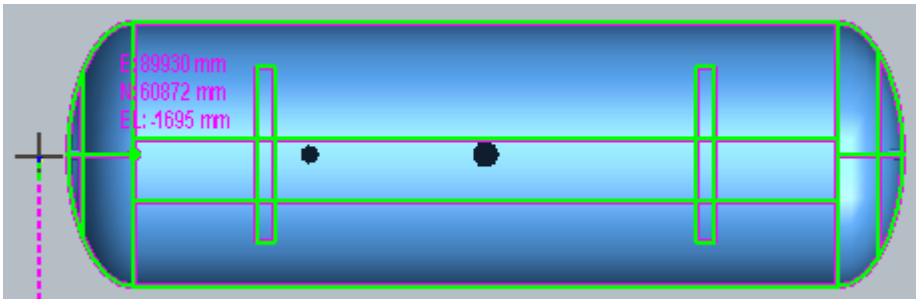
11. If necessary, add or change a positioning relationship by selecting it from the **Positioning Relationships** list.

Define the path for a prismatic shape

1. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.

   TIP If the **Locate Filter** box is not displayed on the horizontal ribbon, click the **Select Tool** on the vertical toolbar.

2. On the **Place Prismatic Shape** ribbon, click **Path** .
3. Click **Create** on the ribbon.
4. Click the first point for the path.

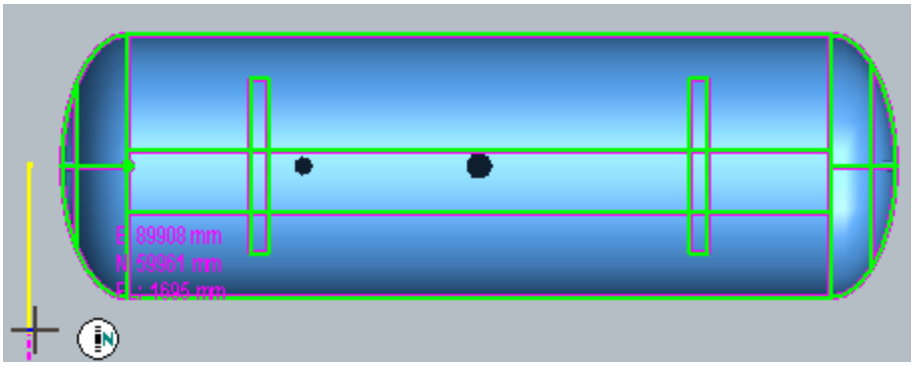


TIPS

- You can use the **PinPoint** or **Point Along** commands and the SmartSketch relationship indicators when defining your path.

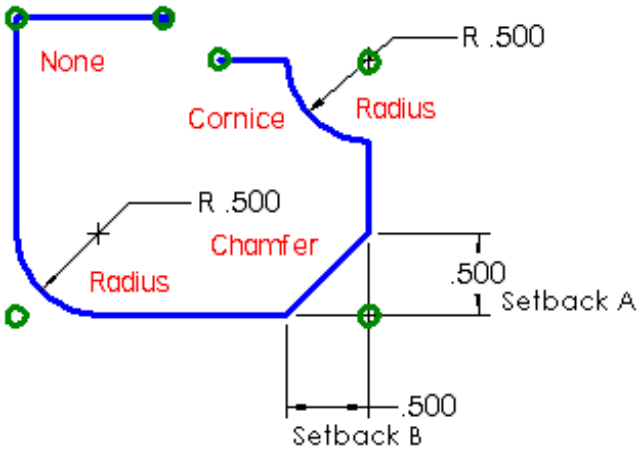
- You can change the plane for the path at any time by selecting the plane in the **Plane** list.
- To change the segment from a straight line to an arc, click **Arc by 3 Points** in the **Path Type** list, and then click three points to define the arc.
- To switch back to a straight line after sketching an arc, click **Line** in the **Path Type** list.
- To break the path, click **No Line** in the **Path Type** list.

5. Click the second point for the path.

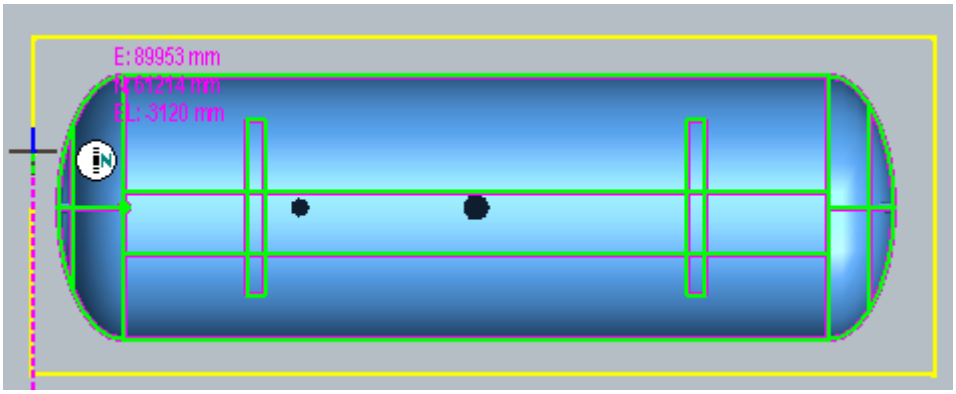


TIP To change the turn type for the corner, click the new turn type in the **Turn Type** list, and define the angle or dimensions for the turn in the **Turn Type Value** box.

The following graphic shows examples of the available turn types:



6. Click to place other segments of the path as needed.



7. After you place all the points that define the path, click **Finish Path**.

NOTES

- The software does not require that you close the path. When creating a continuous path, you can end it at any point.
- After you place a segment of the path by defining two points, you can click **Edit** on the ribbon to change the segment.

Sketch the cross-section for a prismatic shape

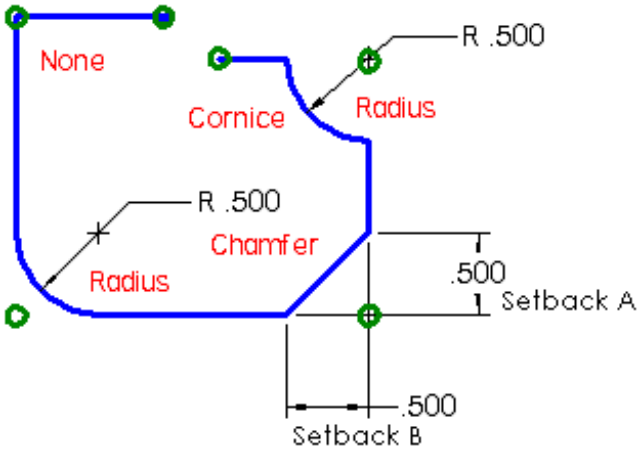
1. On the **Place Prismatic Shape** ribbon, select **Sketch** in the **Cross-Section** list.
2. Click **Cross-Section** .
3. Click the first point for the cross-section.

TIPS

- You must sketch the cross-section on the two-dimensional plane that is orthogonal to the first leg of the path. Click **Show Cross Section** to display a window in the two-dimensional plane. The software displays this plane, which is perpendicular to the path, as you sketch the cross-section.
- The cardinal point, which is the point where the cross-section is attached to the path, is defined as you sketch. In other words, the cross-section surrounds the path and is attached to the path exactly as you sketch it.
- To change the segment from a straight line to an arc, click **Arc by 3 Points** in the **Path Type** list, and then click three points to define the arc.
- To switch back to a straight line after sketching an arc, click **Line** in the **Path Type** list.

4. Click the next point for the cross-section.

TIP To change the turn type for the corner, click the new turn type in the **Turn Type** list and define the angle or dimensions for the turn in the **Turn Type Value** box. The following graphic shows examples of the available turn types:



5. Click to place other segments of the cross-section as needed.
6. Click the starting point of the cross-section to close it.
7. Click **Finish Path**.

   TIP The **Finish Path** button is not available until you close the cross-section by clicking the starting point.

NOTE After you place a segment of the cross-section by defining two points, you can click **Edit** on the ribbon to change the segment.

Define cross-sectional properties for a prismatic shape

1. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.

   TIP If the **Locate Filter** box is not displayed on the horizontal ribbon, click **Select Tool** on the vertical toolbar.

2. On the **Place Prismatic Shape** ribbon, select a cross-section type in the **Cross- Section Type** list.

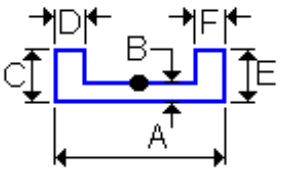
   TIP If you select **Sketch** in the **Cross-Section Type** list, you must sketch the cross-section using the **Path** commands before you can edit properties.

3. Click **Properties** .
4. On the **Prismatic Shape Properties** dialog box, click the **Cross-Section** tab.

5. Define values for each dimension of the cross-section. Letters such as A, B, and C represent the cross- sectional dimensions.

   TIPS

   - To see what each lettered dimension represents on the cross-section, click **Display Cross-Section Image** to view a graphic of the selected cross-sectional type if one exists in the reference data.
   - Cross-sectional images show the dimensions that you can define for the cross-section type, the default cardinal points for the cross-section type, and the angle for the cross-section. For example, the following graphic contains the dimensions and cardinal point for a standard road cross-section.

   Not all cross-section types have images associated with them.

6. Select the cardinal point for the cross-section in the **Cardinality** list.

   TIPS

   - The cardinal point is the point where the software attaches the cross-section to the path.
   - If you click **Display Cross- Section Image** to see a graphic of the cross-section, you can view where each cardinal point is located by selecting each cardinal point in the list.

7. Type an **Angle** for the cross-section, if needed.
8. Click **OK**.

NOTES

- If you sketch a cross-section, the properties for the cross-section include the X-, Y-, and Z-coordinates, the turn type, and values for each point that define the cross-section path.
- You cannot modify the properties of a sketched cross-section on the **Prismatic Shape Properties** dialog box. Instead, you must make changes to the path that defines the cross-section sketch. For more information, see *Modify a Sketched Cross-Section* (on page 353).

Modify a sketched cross-section

1. Click **Select**.
2. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape with a sketched cross-section.
3. On the ribbon, click **Cross-Section**. The software outlines the cross-section path in yellow.
4. To modify a straight segment in the cross-section, select the segment to modify, and then make changes on the ribbon.

   Modify a Straight Segment in a Path (on page 359)

5. To modify an arc in the cross-section, select the arc to modify, and then make changes on the ribbon.

   Modify an Arc in a Path (on page 359)

6. To modify a turn in the cross-section, select the turn to modify, and then make changes on the ribbon.

   Modify a Turn in a Path (on page 360)

7. To move a segment in the cross-section, click the segment to move, and then click the point from which to move the segment and the point to which to move the segment.

   Move Segments of a Path (on page 357)

8. To add segments to the cross-section, click **Create** on the ribbon, and then click to place the new segments.

   Add Segments to a Path (on page 355)

   TIP You can add as many segments to the cross-section as you need. However, you must close the cross-section to be able to save it.

9. To delete a segment in the cross-section, select the segment to delete, and then click **Delete Selected Items** .

10. Click **Finish Path**.

    TIPS

    - The **Finish Path** button is not available until you close the cross-section by clicking the starting point.
    - The new shape appears in dynamics when you click **Finish Path**.
    - The new shape is actually created when you click **Finish** on the **Place Prismatic Shape** ribbon.

Edit cross-section properties for a prismatic shape

1. Click **Select** .
2. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.

   TIP If the **Locate Filter** box is not displayed on the horizontal ribbon, click the **Select Tool** on the vertical toolbar.

3. On the ribbon, click **Properties** .
4. In the **Prismatic Shape Properties** dialog box, click the **Cross-Section** tab.
5. Make modifications to the cross-section properties.

   TIP As you make changes to the cross-section properties, the cross-section changes appear dynamically in the model.

6. Click **OK**.
7. Click **Finish** to apply the changes to the cross-section to the volume.

NOTES

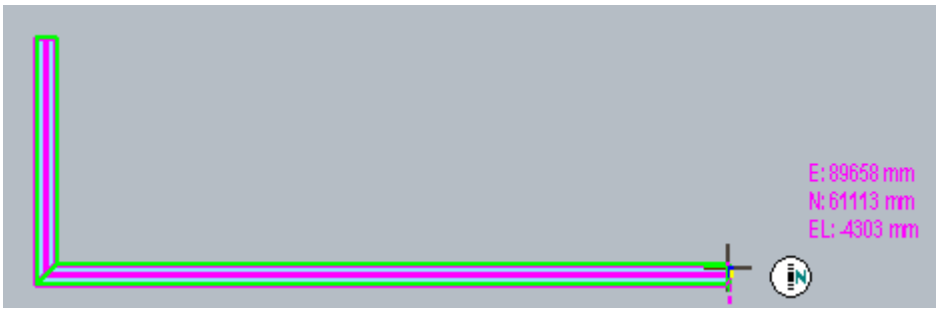
- If you sketch a cross-section, the properties for the cross-section include the X-, Y-, and Z-coordinates, the turn type, and values for each point that define the cross-section path.

- You cannot modify the properties of a sketched cross-section in the **Prismatic Shape Properties** dialog box. Instead, you must make changes to the path that defines the cross-section sketch. For more information, see *Modify a sketched cross-section* (on page 353).

Add segments to a path

1. Click **Select** .
2. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.
3. On the ribbon, click **Path** .

   TIP The software highlights the path.
4. To add one or more segments to the path, click **Create**.
5. Click the point on the existing path to insert the new segment.



TIPS

- You can use **PinPoint**, **Point Along**, and the SmartSketch relationship indicators when defining your path.
- You can change the plane for the path at any time by selecting the plane in the **Plane** list.
- To change the segment from a straight line to an arc, click **Arc by 3 Points** in the **Path Type** list, and then click three points to define the arc.
- To switch back to a straight line after sketching an arc, click **Line** in the **Path Type** list.
- To break the path, click **No Line** in the **Path Type** list.

6. Click to place other points and add to the path as needed.



TIP To change the turn type for the corner, click the new turn type in the **Turn Type** list and define the angle or dimensions for the turn in the **Feature Value** box.

7. After you place all the points for the new segments, click **Finish Path**.



NOTES

- The software does not require that you close the path. You can end the path at any point.
- You can click **Edit** on the ribbon to change the segment or modify the path further.
- To manually set the length and angle for a segment of the path, change the values in the **Angle** and **Length** boxes on the ribbon.

See Also

Creating Customized Shapes (on page 337)

Move Segments of a Path

1. Click **Select** .
2. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.



3. On the ribbon, click **Path** .

   The software highlights the path.

4. Select the segments to move.



   TIP You can select multiple segments by holding the CTRL key and clicking the segments.

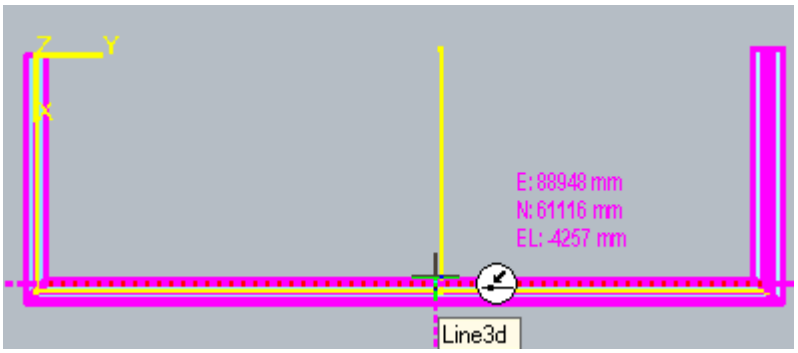
5. To keep the length of a straight segment constant while you move the segment, click **Length Locked** .

   TIPS

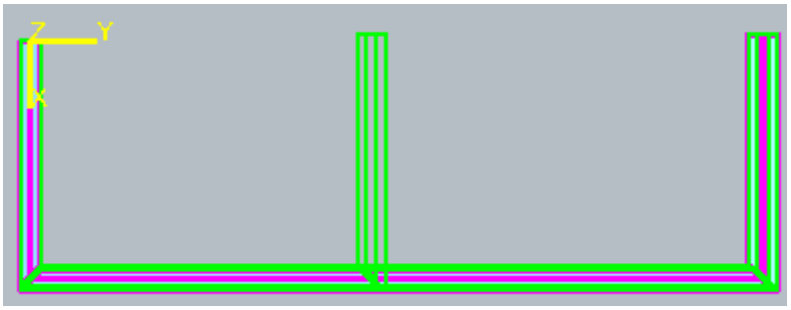
   - When the segment length is locked, the software automatically modifies the turn points, along with the length and angle of the adjacent segment, to remain connected to the moved segment. The length of the moved segment does not change.
   - When the segment length is not locked, the software extends or shortens the associated segments to connect with the new position of the moved segment. The length of the moved segment can change.

6. Click **Move From** to specify the starting location of the move vector.

7. Click to specify the ending location of the move vector.

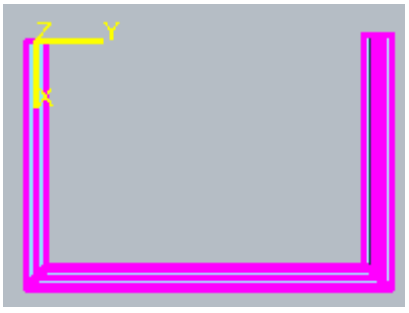


8. Click **Finish Path**.



TIPS

- The new shape appears in dynamics when you click **Finish Path**. The new shape is actually created when you click **Finish** on the **Place Prismatic Shape** ribbon.



- While modifying several elements one at a time, you must use CTRL to select the next element, and then CTRL to de-select the previous element.

Modify a straight segment in a path

1. Click **Select** .
2. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.
3. On the ribbon, click **Path** .

   The software highlights the path.
4. Select the straight segment to modify.

   TIP You can select multiple segments by holding CTRL and clicking the segments.
5. To change the line type for the segment, click a new type in the **Path Type** list.
6. To change the plane for the segment, click a new plane in the **Plane** list.
7. To delete the segment, click **Delete Selected Items** .
8. To keep the length of a straight segment constant when you move the segment, click **Length Locked** .

   TIPS

   - When the segment length is locked, the software automatically modifies the turn points, along with the length and angle of the adjacent segment, to remain connected to the moved segment. The length of the moved segment does not change.
   - When the segment length is not locked, the software extends or shortens the associated segments to connect with the new position of the moved segment. The length of the moved segment can change.
9. Click **Finish Path**.

   TIP The new shape appears in dynamics when you click **Finish Path**. The new shape is actually created when you click **Finish** on the **Place Prismatic Shape** ribbon.

NOTE You can also move the segments of a path. For more information, see *Move Segments of a Path* (on page 357).

Modify an Arc in a Path

1. Click **Select** .
2. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.
3. On the ribbon, click **Path** .

   The software highlights the path.
4. Select the arc to modify.

   TIP You can select multiple segments by holding CTRL and clicking the segments.
5. To change the line type for the segment, click a new type in the **Path Type** list.
6. To change the plane for the segment, click a new plane in the **Plane** list.

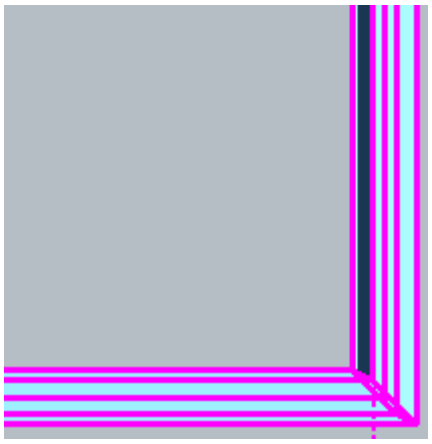
7. To delete the segment, click **Delete Selected Items** .
8. Click **Finish Path**.

   TIP The new shape appears in dynamics when you click **Finish Path**. The new shape is actually created when you click **Finish** on the **Place Prismatic Shape** ribbon.

NOTE You can also move the segments of a path. For more information, see *Move Segments of a Path* (on page 357).

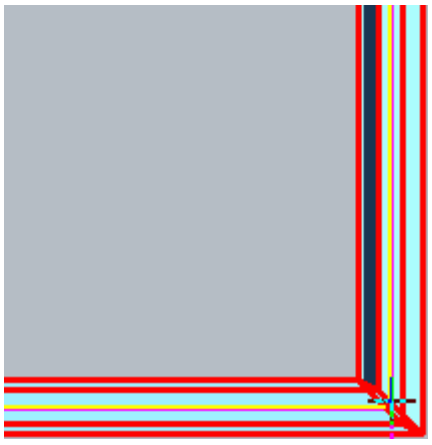
Modify a turn in a path

1. Click **Select** .
2. On the horizontal ribbon, set the **Locate Filter** box to **Shape**, and then select a prismatic shape.



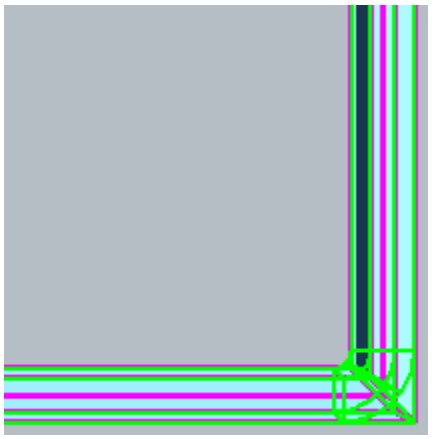
3. On the ribbon, click **Path** .

   The software outlines the path in yellow.
4. Select the turn to modify.

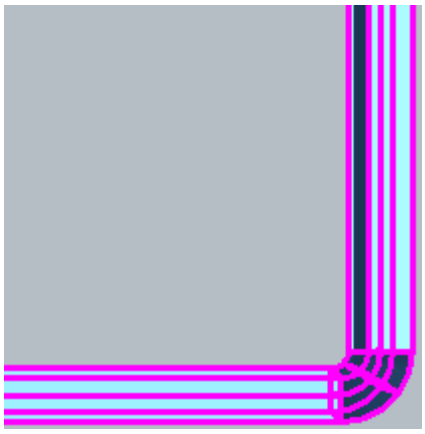


TIP When the pointer passes over an object that contains multiple elements, use the **QuickPick** feature to help you select the specific element.

5. To change the plane for the segment, click a new plane in the **Plane** list.
6. To delete the turn, click **Delete Selected Items** .
7. To change the turn type, select a new type in the **Turn Type** list.
8. To change the dimensions for the selected turn type, type a value in the **Feature Value** box.
9. Click **Finish Path**.



TIP The new shape appears in dynamics when you click **Finish Path**. The new shape is actually created when you click **Finish** on the **Place Prismatic Shape** ribbon.



NOTE You can also move the segments of a path. For more information, see *Move Segments of a Path* (on page 357).

Sketch Properties Dialog Box

Sets options for paths. You cannot edit the different applicable properties.

See Also

General Tab (Sketch Properties Dialog Box) (on page 362)

General Tab (Sketch Properties Dialog Box)

Displays the points that make up a path, their X-, Y-, and Z-coordinates, turn types, and turn type dimensions.

Point No

Displays the point number that identifies the selected point.

X

Displays the location of the point on the X-axis.

Y

Displays the location of the point on the Y-axis.

Z

Displays the location of the point on the Z-axis.

Turn Type

Displays the type of turn associated with the point. Turn types include none, bend, chamfer, and cornice.

Value

Specifies dimensions for the selected turn type. For bends and cornices, the value specifies the radius of the bend. For chamfers, the value specifies the dimensions for setback A and setback B of the chamfer.

SECTION 23

Place Imported Shape from File

Adds geometry to a designed equipment object that was modeled with solid modeling software and saved to a VUE, SAT, or MicroStation DGN file format to an equipment or equipment component object. The **Place Imported Shape from File** command uses the equipment or equipment component object selected in the **Workspace Explorer** hierarchy. If no equipment or equipment component object is selected prior to selecting the command, you are prompted to select an equipment or equipment component object.

The command creates an *Imported Shape* object with the defined properties and a local coordinate system. The imported shape coordinate system is defined by the VUE, SAT, or MicroStation file. The new imported shape does not retain a connection to the original import file.

After importing a shape, you can use the **Place Nozzle** command to complete the equipment definition. For more information, see Place Nozzle in the *Equipment and Furnishings User's Guide*.

You can also edit the shape properties. For more information, see *Edit Shape Properties* (on page 332).

NOTE The **Place Imported Shape from File** command supports SAT files of ACIS R22 and earlier.

Select Shape File Dialog Box

Specifies the VUE, SAT, or MicroStation DGN file that contains the geometry to import as the shape. This dialog box appears automatically when you click the **Place Imported Shape from File** command. You can navigate through the available file systems to locate the appropriate file. After you select a file and click **OK**, the **Display Aspects** dialog box appears so you can select the aspect(s) for the imported shape. The software returns you to the model to place the imported shape.

When you use **Place Imported Shape from File** to place an imported shape from a .sat, .dgn, or .vue file on the symbol share, Smart 3D remembers the relative path to the file. This lets you share design files between models during **Copy**/**Paste** operations, **Model Data Reuse**, **Paste from Catalog**, and **Mirror Copy** of imported shapes. If you import a shape file from a location other than the symbol share, Smart 3D remembers the full path to that file. In both cases, the full path displays on the **Properties** dialog box for the shapes. For more information, see *Share imported shapes between models* (on page 366).

NOTE To save the relative path to imported shapes placed with earlier versions of Smart 3D, you must remodel those shapes.

You can also right-click the new shape object and select **Properties** to further define the object. Use **Place Nozzle** to add nozzles and/or ports to the object.

NOTE The **Select Shape File** dialog box is a standard Microsoft Windows dialog box. For information regarding how to use the dialog box, click the question mark **?** in the upper right corner, and then click a control on the dialog box.

Place an Imported Shape

1. Click **Place Imported Shape from File** on the vertical toolbar.

   TIP If an equipment or equipment component object has not been selected, select one in a graphic view or in the **Workspace Explorer**.

   *The **Select Shape File** dialog box appears.*

2. Navigate to and select the VUE, SAT, or MicroStation DGN file that contains the shape to insert, and then click **OK**.

3. In the **Display Aspect** list box, select the aspects from the list.

   TIP You can click **Format** > **View** and change the **Render Selected Aspects** option in the **Format View** dialog box to display the aspect in the active graphic view.

4. Click **OK**.

   The list box closes, and you return to the model.

5. Click in the graphic view to select an approximate location or reference element for a relationship for the shape.

NOTE After a shape has been placed in the model as part of the equipment or equipment component object, you can use the horizontal ribbon to add or change the positioning relationship. You can add port definitions with the **Place Nozzle** command to complete the equipment or equipment component definition.

SAT File Limitations

- From a performance point of view, the SAT file size should be less than 5 MB.
- From a drawings point of view, the files must not be detailed and must not spread across a large space as they are caught in volume filters for Drawings.
- Spreading the SAT files across multiple equipment reduces the volume of each .sat file.
- For efficient Interference Checker processing, SAT files should not have a footprint (range) greater than 100 meters.
- When we import the SAT files created using a third-party tools, they are converted to Intergraph GType format. In some cases, the generated objects may not be valid after the dual conversion, such as when you use AutoCAD to convert an object to ACIS SAT format, and then convert it to GType. The possible causes for this failure are:
  - You are using an old version of the SAT file.
  - The units of measurement do not match. For example, scaling can result in a degenerate geometry if the distance between two points becomes less than 1.0E-6 M.
  - The orientation of the boundary curves is inconsistent with the surface normal. For example, when applying the right-hand rule, the object is on your left when moving in the direction on the boundary, and the surface normal is upward.
- The valid geometries that you can use in creating SAT files are listed below.
  - GLineTYPE
  - GEllipseTYPE

- GArcTYPE
- GLineStringTYPE
- GBspCurveTYPE
- GComplexStringTYPE
- GBspSurfaceTYPE
- GTorusTYPE
- GSphereTYPE
- GConeTYPE
- GTippyConeTYPE
- GPlaneTYPE
- GShapeTYPE
- GProjectionTYPE
- GPipeTYPE
- GRevolutionTYPE
- GElbowTYPE
- GRuledTYPE
- GRuledNormalTYPE
- GValveTYPE
- GReducerTYPE
- GPolyMeshTYPE
- GPointTYPE
- GVolumeTYPE
- GText3dTYPE

VUE File Limitations

- You cannot import the shape from the VUE file under a design solid.
- The VUE file size should be less than 10 MB. If the VUE file size exceeds 10 MB, the software displays a warning message.
- If the VUE file size exceeds 25 MB, the software displays an error message and you cannot place the object.
- The imported graphics are not smart graphics. They do not contain properties.
- The valid geometries that you can use are the same as those you can use in creating SAT files.

Share imported shapes between models

For imported shapes placed from a file on the symbol share, Smart 3D remembers the relative path to that file. If you import a shape file from a location other than the symbol share, Smart 3D remembers the full path to that file. In both cases, the full path displays on the **Properties** dialog box for the shapes.

When you place these shapes in the model using a file on the symbol share, and then later use **Copy**/**Paste** operations, **Model Data Reuse**, **Paste from Catalog**, or **Mirror Copy**, then the shapes will refer to a file on the target model symbol share rather than the source model symbol share. If the shapes in the source model are placed using a file from somewhere other than the symbol share, then during the **Copy**/**Paste** operations, **Model Data Reuse**, **Paste from Catalog**, or **Mirror Copy**, the shapes will refer to that physical path.

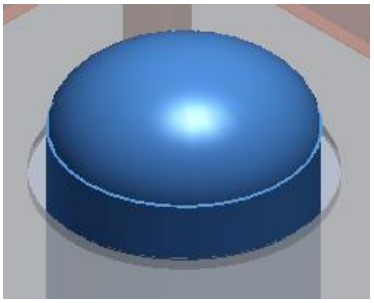
NOTES

- During **Model Data Reuse**, **Copy/Paste**, or **Paste from Catalog**, the software verifies that the target file exists. If the target file does not exist in the path, Smart 3D does the following:
  - Copies the graphics of the source model imported shape.
  - Pastes the same graphics in the target model imported shape.
  - Adds an item to the **To Do List** indicating that the symbol failed because the imported shape source file is not found in the path. This applies to shapes imported from shared content as well as shapes imported from a physical location.
- If the software detects a problem during **Model Data Reuse**, **Copy/Paste**, or **Paste from Catalog**, such as an empty file or a file that it cannot convert, then Smart 3D does the following:
  - Copies the graphics of the source model imported shape.
  - Pastes the same graphics in the target model imported shape.
  - Adds an item to the **To Do List** with an appropriate message.

SECTION 24

Place Opening

Places an opening (hole) in a slab, wall, or member created in the Structure or Civil task. The shape of the opening can be defined by placing a pre-defined shape from the catalog, sketching the opening outline, or defining the boundaries for the opening. In addition, you can control the depth of the opening to create a fully penetrating hole or a recessed opening.



Place Opening Ribbon

Specifies the properties for the opening that you are placing or editing.



Opening Properties

Activates the **Opening Properties** dialog box. You can use this dialog box to specify opening properties that you cannot set on the ribbon. For more information, see *Opening Properties Dialog Box* (on page 381).

Structure

Select the object in which to place an opening. You can select members or slabs.

Sketching Plane

Select the plane on the object on which to draw the opening. If you are placing an opening that does not go all the way through the object, the sketching plane defines the side of the object in which the partial opening is placed.

NOTE If you are editing an opening in the Structure or Civil task, this option is only available if the sketching plane for the opening is missing.

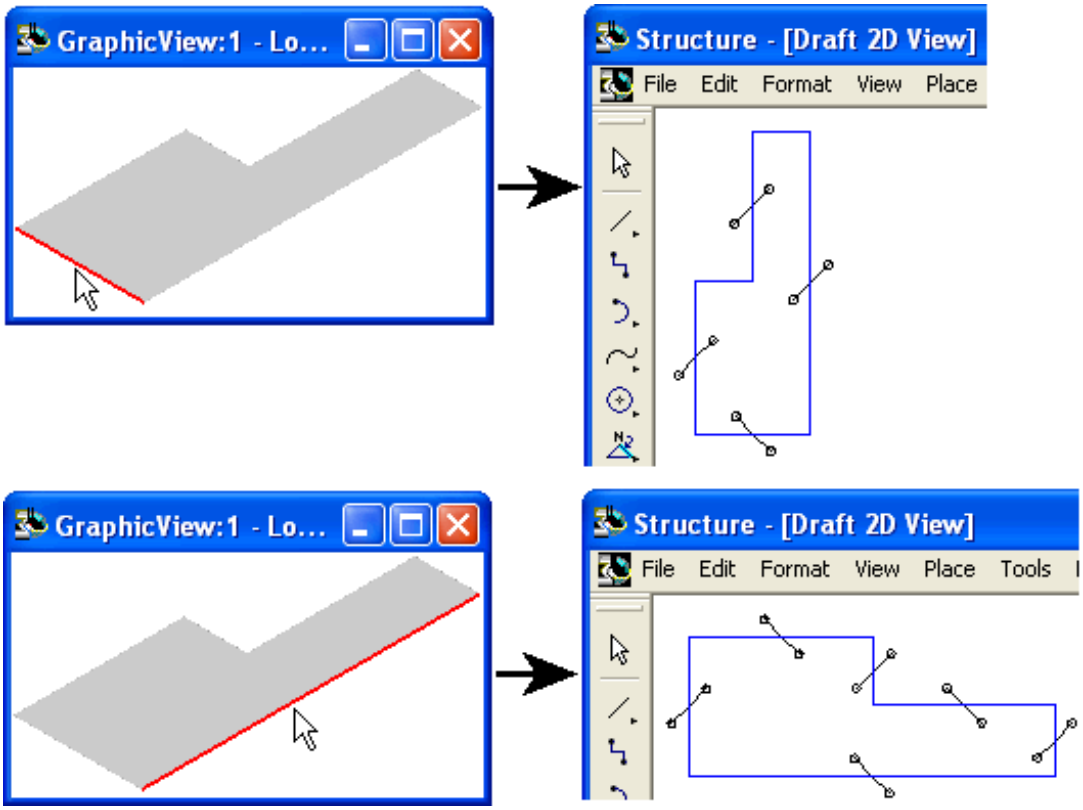
Define Orientation

Select this option if you plan to sketch the opening and want to define the orientation of the object, in which the opening is being placed, in the 2D environment. Most of the time, the default orientation that the software uses is appropriate for the opening. However, use this option if you are placing an opening in a multi- sided slab, for example, and you need to know that the correct side of the slab is selected for the opening.

Edge for 2D X-axis

Select this option to specify the object edge that should be oriented along the bottom of the 2D environment. This option is only available when you select the **Define Orientation**

option.



End Point

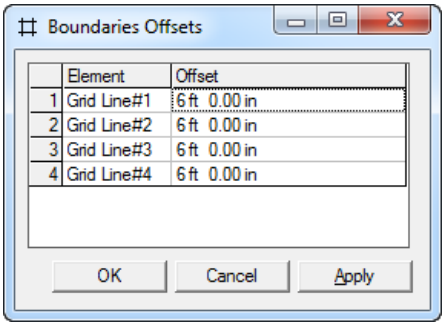
Select this option to specify the lower left corner of the object as you want it to appear in the 2D environment. This option is only available when you select the **Define Orientation** option.

Boundary

Specify the boundaries of the opening. If you select objects in the model and those objects are moved, the software automatically resizes the opening to maintain the boundary relationship. For example, you can select the outside edge of a beam flange as a boundary along one side of an opening. Later, if the section size for the beam is changed and the flange edge moves out an inch, the software automatically moves the edge of the opening an inch to match the new location of the flange edge.

Boundary List

Activates the **Boundary Offsets** dialog box. This dialog box allows you to define an offset distance for each boundary that you have defined. Select a row to highlight the boundary in the model. Then, type the distance from the boundary to place the edge of the opening. Type 0 to place the opening edge on the boundary. Type a negative number to offset the opening inside the boundary. Type a positive number



to offset the opening outside the boundary.

Finish

Places the opening using the defined parameters.

Cancel

Rejects the objects that you have selected.

Accept

Confirms the objects that you have selected.

Select Boundaries

Select this option to select objects in the model to define the boundaries of the opening. This option is only available when you are specifying boundaries.

Add References to Sketch 2D

Adds an object as a reference object in the 2-D environment. The objects that you select will display as thick blue lines in the 2-D environment. You can define relationships and dimensions to these blue-lined objects in 2-D. The software may also automatically retrieve 3-D objects to display in the 2-D environment. The automatically selected objects will display as grey lines. These grey line objects are read-only (you cannot define relationships or dimensions to them) and are there to help you visualize the model. This option is only available when you are drawing the boundaries of the opening.

NOTE For piping and equipment objects in plant mode, **Insulation**, **Equipment Hole**, **Maintenance**, and **Operation** aspects displaying in the 3D graphic view also display in the Sketch 2D environment. The aspect geometry displays as dashed lines. You can define dimensions, but not relationships, to these aspects in 2D.

TIP The software also automatically retrieves 3-D objects to display in the 2-D environment. The automatically selected objects display as fine grey lines. Examples of these objects include geometry generated by end cut operations, geometry that represents hidden lines from intersecting objects projected to the sketching plane, or objects that intersect the sketching plane. The software writes these graphics to the No_Constrained_Elements layer. You can reference these objects to place your sketch, but the objects that display as fine grey lines are temporary. When you exit the 2-D environment, the software deletes these graphics. Any relationships or constraints to the temporary graphics are also deleted when you exit the 2-D environment.

Although the references are deleted when you exit the 2D environment, the references are automatically reimported when you modify the opening.

Draw

Activates the Structure Draft 2D View environment. Use this environment to sketch the outline of the opening in the structure that you have selected.

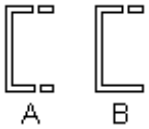
Shape

Specifies how you want to define the opening shape. Select **Sketch** to draw the opening shape. Select **More** to select an opening shape from the catalog. This option is only used when you are not using boundaries to define the opening shape.

Cutting Limit

Select the cutting depth method for the opening.

- Select **Through-All** to completely penetrate the object in which the opening is placed. This option is available for all objects in which you can place an opening.
- Select **Through-Next** to penetrate only one side of an object. For example, a channel section member in which you want an opening only on the top flange. This option is available only when the object that you select for the opening is a member whose cross-section has parallel webs or flanges or a wall that turns back on itself. In the figure, A is the **Through-All** option and B is the **Through-Next** option.



- Select **User Defined** if you want to specify the penetration depth of the opening. This option is available for all objects in which you can place an opening.

Cutting Depth

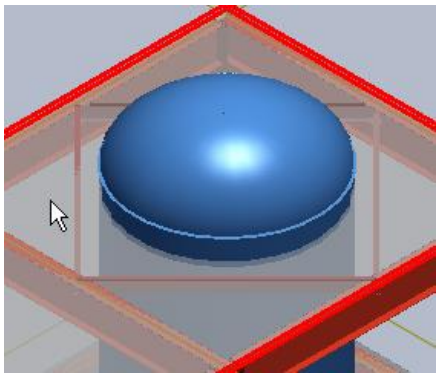
Specify the cutting depth for the opening. The depth is measured from the sketching plane that you specified.

What do you want to do?

- *Place an opening by boundaries* (on page 371)
- *Place an opening by shape* (on page 372)
- *Place an opening by drawing* (on page 374)
- *Place an opening on a sloped wall* (on page 376)
- *Place an opening on a leaning wall* (on page 378)
- *Change a recess opening to a through opening* (on page 380)
- *Change a through opening to a recess opening* (on page 380)
- *Delete an opening* (on page 380)

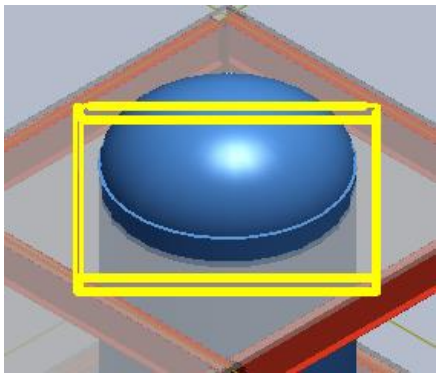
Place an opening by boundaries

1. Click **Place Opening** on the vertical toolbar.
2. Select the object in which you want to place the opening. You can select walls, slabs, and members.

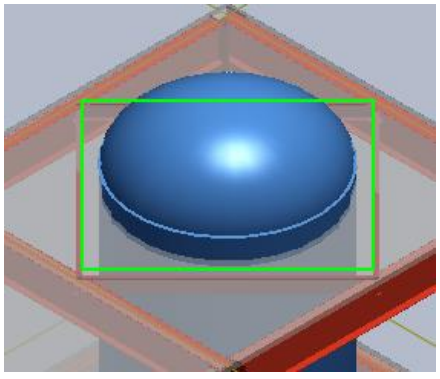


3. Click **Select Boundaries**.
4. Select objects in the model to use as boundaries of the opening.

   NOTE You must select the boundary objects in a clockwise or counterclockwise order.



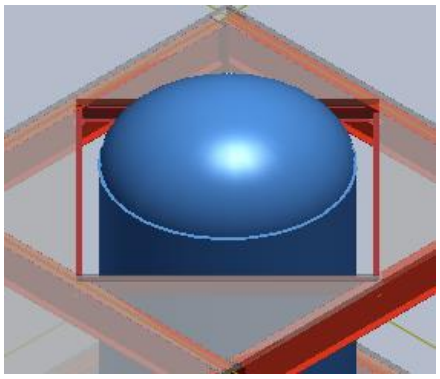
5. Click **Accept**.



6. Optionally, define any offset from the boundaries that you have selected.

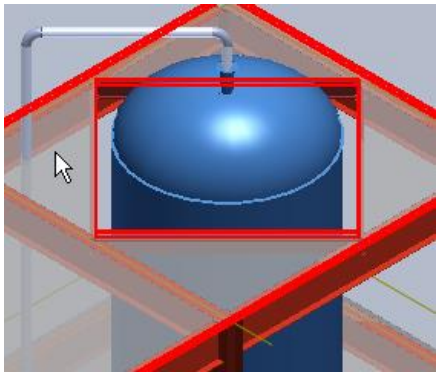
TIP Type 0 to place the opening edge on the boundary. Type a negative number to offset the opening inside the boundary. Type a positive number to offset the opening outside the boundary. This offset can only be applied when the selected boundaries are lines or planes.

7. Click **OK** on the **Boundary** dialog box.
8. Click **Finish**.



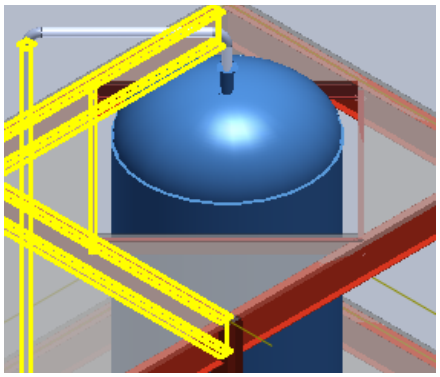
Place an opening by shape

1. Click **Place Opening** on the vertical toolbar.
2. Select the object in which to place the opening. You can select walls, slabs, and members.

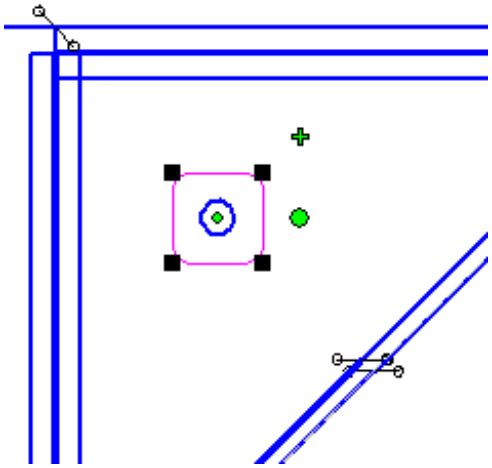


3. Click **Add References to Sketch 2D** .

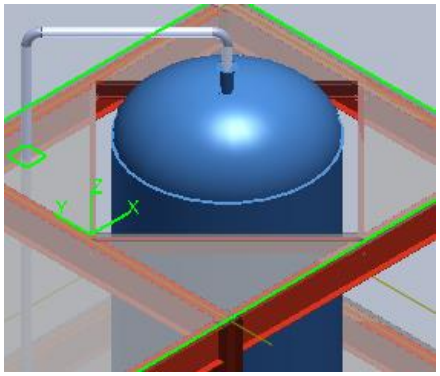
4. Select objects that you need to reference when placing the opening.



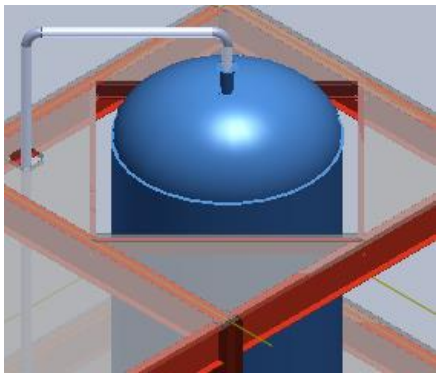
5. In the **Shape** option, select **More...**.
6. Select the catalog shape that you want to place, and then click **OK**.
7. Drag the shape to where you want the opening on the object.



8. Click **Close** on the ribbon bar.

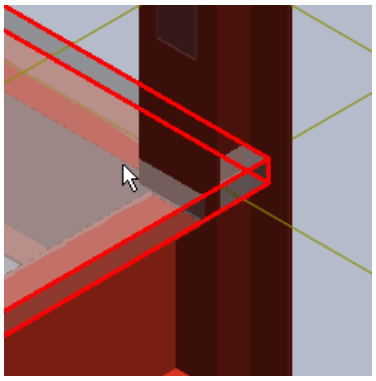


9. Click **Finish**.



Place an opening by drawing

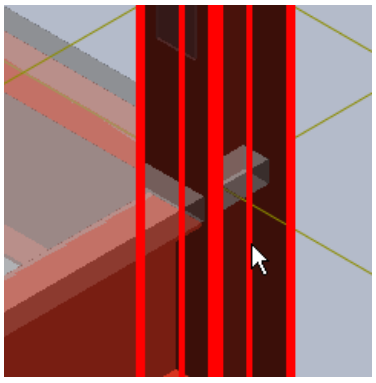
1. Click **Place Opening** on the vertical toolbar.
2. Select the object in which to place the opening. You can select walls, slabs, and members.



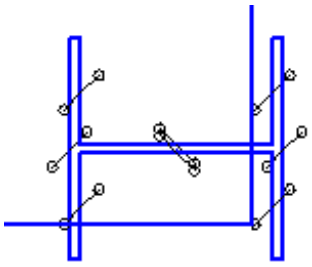
3. Click **Add References to Sketch 2D**.
4. Select objects in the 3-D model near the object in which you are placing the opening. The objects that you select display as thick blue lines in the 2-D environment. You can define relationships and dimensions to these blue-lined objects in 2-D.

   TIP The software also automatically retrieves 3-D objects to display in the 2-D environment. The automatically selected objects display as fine grey lines. Examples of these objects include geometry generated by end cut operations, geometry that represents hidden lines from intersecting objects projected to the sketching plane, or objects that intersect the sketching plane. The software writes these graphics to the No_Constrained_Elements layer. You can reference these objects to place your sketch, but the objects that display as fine grey lines are temporary. When you exit the 2-D environment, the software deletes these graphics. Any relationships or constraints to the temporary graphics are also deleted when you exit the 2-D environment.

Although the references are deleted when you exit the 2D environment, the references are automatically reimported when you modify the opening.

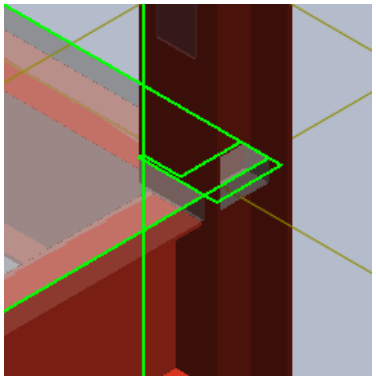


5. Click **Draw** .

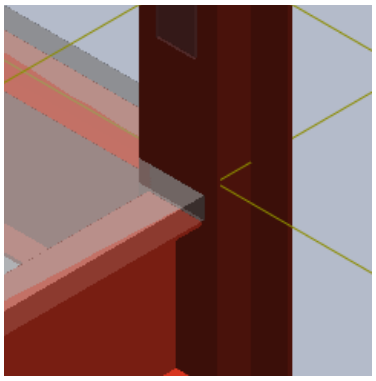


6. Using the available drawing commands in the 2-D environment, draw the opening.
7. Click **Close** on the ribbon bar.

   The opening that you drew displays in the 3-D environment.



8. Click **Finish**.



Place an opening on a sloped wall

Hole Parallel with Wall Slope

1. Click **Place Opening** on the vertical toolbar.
2. Select the sloped wall.
3. Select the side of the wall in which to place the opening.

   TIP If you are placing an opening that completely penetrates the wall, the side selection is not important. However, if you are placing an opening that does not completely penetrate the wall, then the surface selection is important because the penetration depth is measured from the selected surface.

4. Click **Draw**.

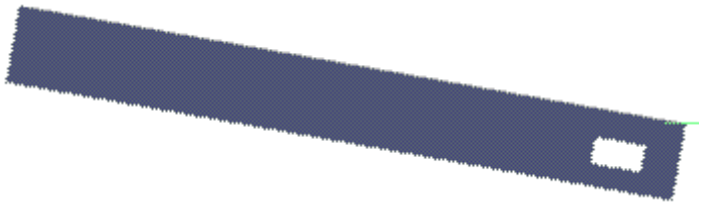
   TIP Regardless of the slope, the wall displays as level in the 2-D environment.

5. Using the available drawing commands in the 2-D environment, draw the opening. In the example below, a small rectangular hole was drawn at the wall end.



6. Click **Close** on the ribbon bar.
7. Specify the cutting limit to use.
8. Click **Finish**.

   The software places the hole in the sloped wall parallel to the slope.



Hole Parallel with Another Model Object

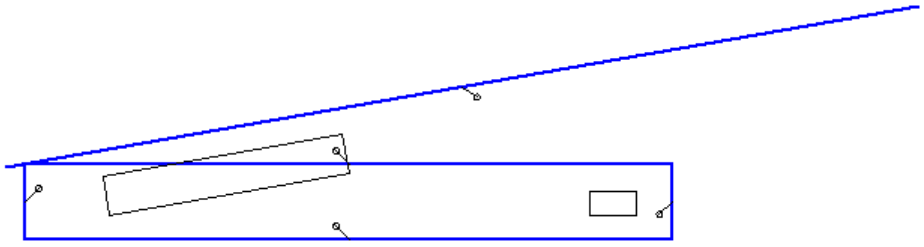
1. Click **Place Opening** on the vertical toolbar.
2. Select the sloped wall.
3. Select the side of the wall in which to place the opening.

   TIP If you are placing an opening that completely penetrates the wall, the side selection is not important. However, if you are placing an opening that does not completely penetrate the wall, then the surface selection is important because the penetration depth is measured from the selected surface.
4. Click **Add References to Sketch 2D**.
5. Select an object in the 3-D model near the wall that you want the hole parallel to. For example, select a grid line to make the hole parallel to a plane. The objects that you select will display as thick blue lines in the 2-D environment. You can define relationships and dimensions to these blue-lined objects in 2-D. The software may also automatically retrieve 3-D objects to display in the 2-D environment. The automatically selected objects will display as grey lines. These grey line objects are read-only (you cannot define relationships or dimensions to them) and are there to help you visualize the model.

   TIP The software also automatically retrieves 3-D objects to display in the 2-D environment. The automatically selected objects display as fine grey lines. Examples of these objects include geometry generated by end cut operations, geometry that represents hidden lines from intersecting objects projected to the sketching plane, or objects that intersect the sketching plane. The software writes these graphics to the No_Constrained_Elements layer. You can reference these objects to place your sketch, but the objects that display as fine grey lines are temporary. When you exit the 2-D environment, the software deletes these graphics. Any relationships or constraints to the temporary graphics are also deleted when you exit the 2-D environment.

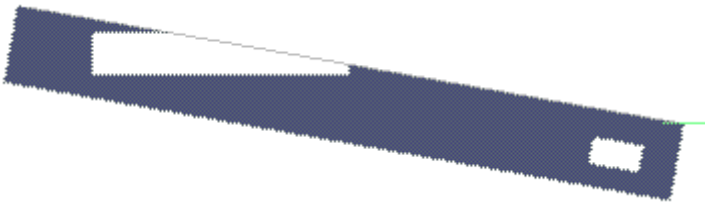
   Although the references are deleted when you exit the 2D environment, the references are automatically reimported when you modify the opening.
6. Click **Draw**.

   The wall and the selected object displays in the 2-D environment.
7. Using the available drawing commands in the 2-D environment, draw the opening. In the example below, the opening was placed using the parallel SmartSketch relationship with a grid line that was selected as a reference.



8. Click **Close** on the ribbon bar.
9. Specify the cutting limit to use.
10. Click **Finish**.

The software places the hole in the sloped wall parallel to the referenced grid line.

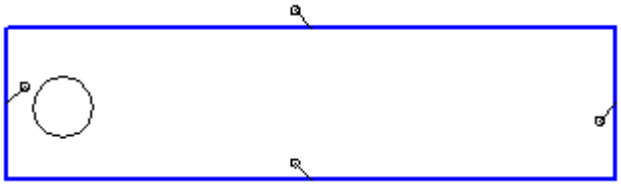


Place an opening on a leaning wall

Opening Normal to Side of Wall

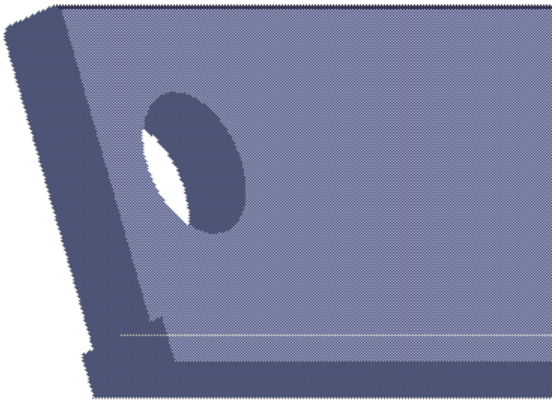
1. Click **Place Opening** on the vertical toolbar.
2. Select the wall.
3. Select the side of the wall in which to place the opening.

   TIP If you are placing an opening that completely penetrates the wall, the side selection is not important. However, if you are placing an opening that does not completely penetrate the wall, then the surface selection is important because the penetration depth is measured from the selected surface.
4. Click **Draw**.
5. Using the available drawing commands in the 2-D environment, draw the opening. In the example below, a circular hole was drawn at the wall end.



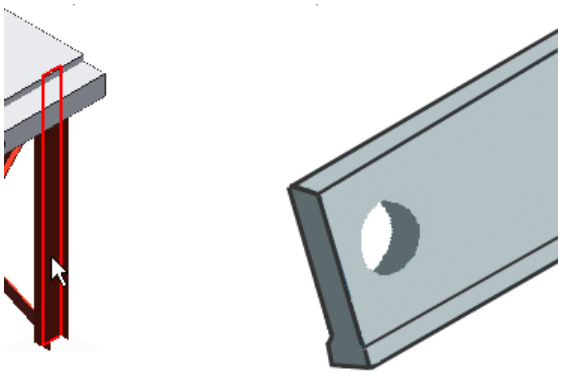
6. Click **Close** on the ribbon bar.
7. Specify the cutting limit to use.
8. Click **Finish**.

The software places the opening normal to the wall face.



Opening Normal to Another Model Object

1. Click **Place Opening** on the vertical toolbar.
2. Select the wall.
3. Select a plane that is normal to the opening that you want. In the example below, the web of a nearby column is selected.

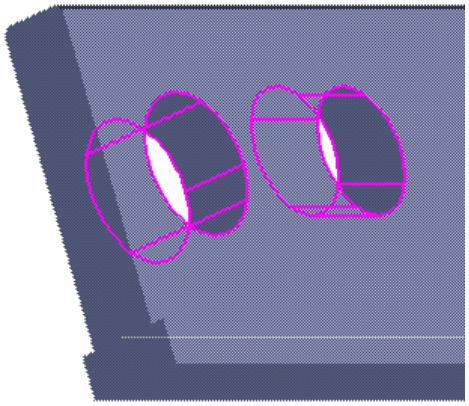


4. Click **Draw**.

   The wall displays in the 2-D environment.
5. Using the available drawing commands in the 2-D environment, draw the opening.
6. Click **Close** on the ribbon bar.
7. Specify the cutting limit to use.
8. Click **Finish**.

   The software places the opening normal to the column's web (right opening), not the wall

face (left opening).



Change a recess opening to a through opening

1. Click **Select** on the vertical toolbar.
2. Select **Openings** in the **Locate Filter**.
3. Select the recess opening to change to a through opening.
4. In the **Cutting Limit** option, select **Through-All**.

Change a through opening to a recess opening

1. Click **Select** on the vertical toolbar.
2. Select **Openings** in the **Locate Filter**.
3. Select the through opening to change to a recess opening.
4. In the **Cutting Limit** option, select **User Defined**.
5. In the **Cutting Depth** option, specify the new depth of the opening.

Delete an opening

1. Click **Select** on the vertical toolbar.
2. Select **Openings** in the **Locate Filter**.
3. Select the opening to delete.
4. Click **Delete** .

Opening Properties Dialog Box

Specifies the properties for the opening that you are editing.

See Also

General Tab (Opening Properties Dialog Box) (on page 381)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

General Tab (Opening Properties Dialog Box)

The **General** tab displays the opening properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one opening and then selected the properties command, only the common properties between the selected openings display.

When viewing properties for a single opening, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on properties. You can access this document using the **Help > Printable Guides** command in the software.

Category - Select the properties that you want to view for the opening. Opening properties have one category: Standard. You select the category to define values for by using the **Category** option.

Standard

The standard properties that display depend on the reference data for the opening. Because opening reference data is fully customizable, only the common properties are documented.

Name

Specify a name for the opening.

Naming Rule

Select the naming rule to use to name the opening.

- **Default Name Rule** - Names the opening using the format "Opening-<location>-<index>" where <location> is the global workshare location ID and <index> is a unique index number that starts at 0001. For example, Opening-1-0043.
- **Unique Name Rule** - Names the wall using the format "Opening-<location>-<index>" where <location> is the global workshare location ID and <index> is a unique index number that starts at 0001. For example, Opening-1-0043.
- **User Defined** - Select to specify the wall system name yourself in the **Name** box.

Description

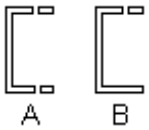
Type a description for the opening.

Cutting Limit

Select the cutting depth method for the opening.

- Select **Through-All** to completely penetrate the object in which the opening is placed. This option is available for all objects in which you can place an opening.

- Select **Through-Next** to penetrate only one side of an object. For example, a channel section member in which you want an opening only on the top flange. This option is available only when the object that you select for the opening is a member whose cross-section has parallel webs or flanges or a wall that turns back on itself. In the figure, A is the **Through-All** option and B is the **Through-Next** option.



- Select **User Defined** if you want to specify the penetration depth of the opening. This option is available for all objects in which you can place an opening.

Cutting Depth

Specify the cutting depth for the opening. The depth is measured from the sketching plane that you specified.

Default constant offset

Specify the offset between the opening edge and the boundary. Type 0 to place the opening edge on the boundary. Type a negative number to offset the opening inside the boundary. Type a positive number to offset the opening outside the boundary. This offset can only be set when the boundaries are lines or planes.

See Also

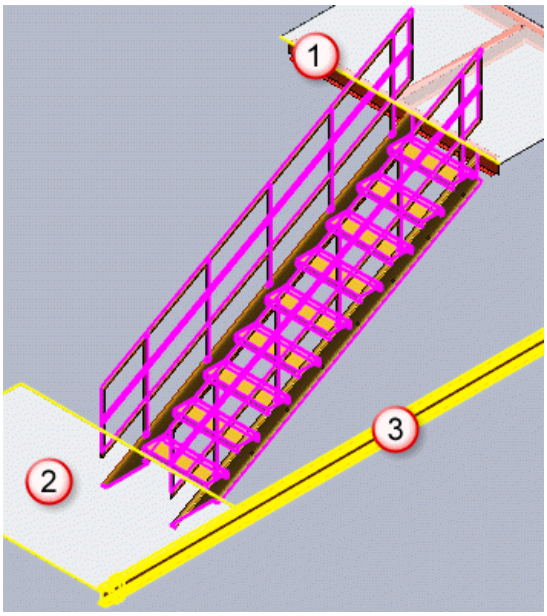
Opening Properties Dialog Box (on page 381)

SECTION 25

Place Stairs

Places a stair in the model. You attach the top of the stair to an edge of any object. You attach the bottom of the stair to a plane. The bottom plane must be lower than the top edge. You must also define the position of the stair along the top edge. There are two methods for defining this position: 1) Select a reference edge or plane that intersects with the top edge and set the horizontal offset distance along the top edge from the intersection point. You can position the stair on either side of the reference. 2) Define a position for the stair graphically.

After you define the position of the stair, the stair appears in the model. The software automatically calculates the number of steps.



1 - Top Edge

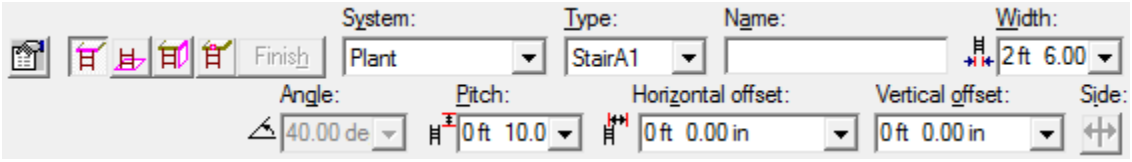
2 - Bottom Plane

3- Reference Edge

NOTE Because the stair, the top edge, and the bottom plane are connected, the interference checker service will not find any hard interference between any part of the stair (stringer, handrail, tread, and so forth) and the top edge object or the bottom plane.

Place Stair Ribbon

Specifies the properties for the stair that you are placing or editing.



Stair Properties

Activates the **Stair Properties** dialog box. You can use this dialog box to specify additional stair properties, such as material and material grade, which you cannot set on the ribbon. For more information, see *Stair Properties Dialog Box* (on page 389).

Select Top Edge

Select the reference plane or edge that defines the top elevation of the stair. If this edge is deleted after the stair has been placed, the software puts the stair on the To Do List.

Select Bottom Plane

Select the reference plane that defines the bottom elevation of the stair. This plane must be lower than the top edge that you selected. The reference plane can be flat or sloped up to 45 degrees. If this plane is deleted after the stair has been placed, the software puts the stair on the To Do List.

Select Reference Edge

Select a plane or edge from which to place the stair along the top edge. This input is optional.

Select Position

Graphically define the location of the stair along the selected top edge.

Finish

Places the stair in the model.

System

Select the system to which the stair belongs. You can create new systems in the Systems and Specifications task.

Type

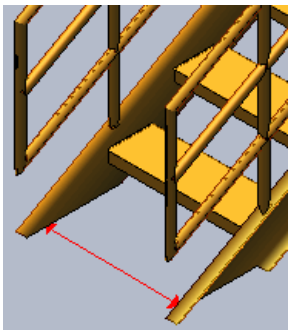
Select the type of stair to place.

Name

Specify the name for the stair that you are placing.

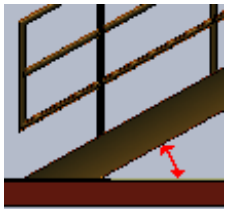
Width

Enter the width of the stair treads. This dimension does not include the width of the stringers and therefore is less than the overall width of the stairs.



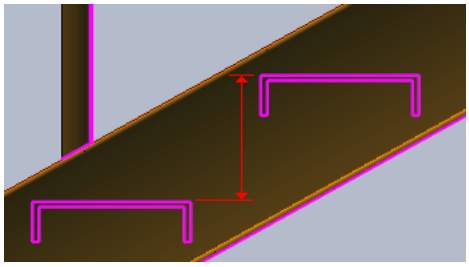
Angle

Enter the angle of inclination between the selected bottom plane and the stair stringers.



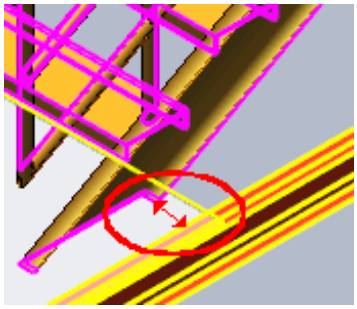
Pitch

Enter the maximum allowable vertical distance between the stair treads. The actual distance is calculated by the software using the vertical distance between the top edge and the bottom plane and the stair angle. The actual distance is displayed in **Actual Step Pitch**.



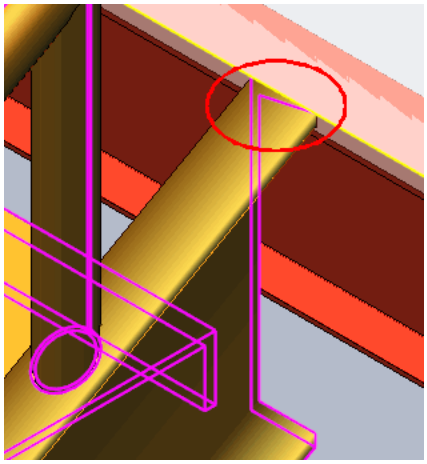
Horizontal Offset

Specify the distance between the outside edge of the stair and the selected reference edge or object.



Vertical Offset

Specify the offset between the top of the stair stringer and the selected top edge.



Side

Places the stair on the opposite side of the top edge.

Select Stair Dialog Box

Allows selection of the type of stairs to be placed. This dialog box appears when you select **More** in the **Type** list. By browsing through the hierarchy, you can find any stair in the Catalog database. After you select a stair, the software returns you to the model, where you can finalize placement.

Back

Returns you to the previously selected stair type or node. Use this command to navigate through the hierarchy to the specific type that you need.

Forward

Sends you to the last selected stair type or node that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific type that you need.

Up One Level

Brings up the next highest level of the catalog hierarchy. Use this command to navigate through the hierarchy to the specific type that you need.

Properties

Displays the properties of the selected stair. Because you cannot modify any properties until the stair is placed, all properties on the dialog box are read-only.

Preview

Displays a picture of the selected stair. The image file must be assigned to the stair in the reference data.

List View

Sets the dialog box to display stairs in a list view.

Grid View

Sets the dialog box to display stairs in a spreadsheet-style grid view.

Address

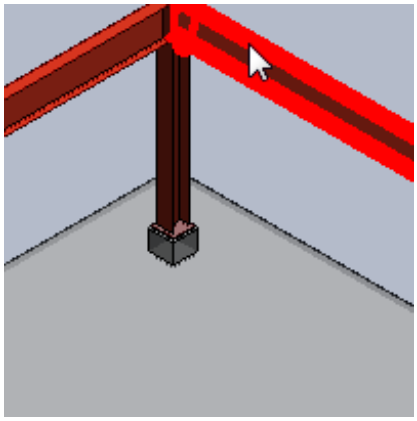
Specifies your exact location within the displayed hierarchy.

What do you want to do?

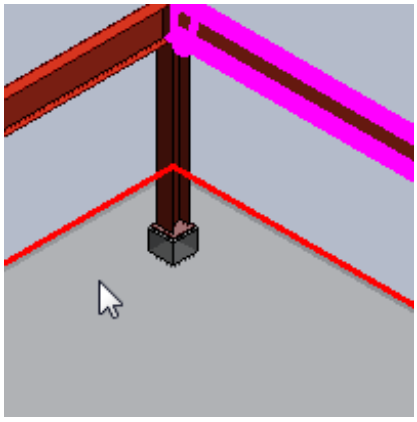
- *Place a stair* (on page 387)
- *Edit stair placement* (on page 389)
- *Delete a stair* (on page 389)

Place a stair

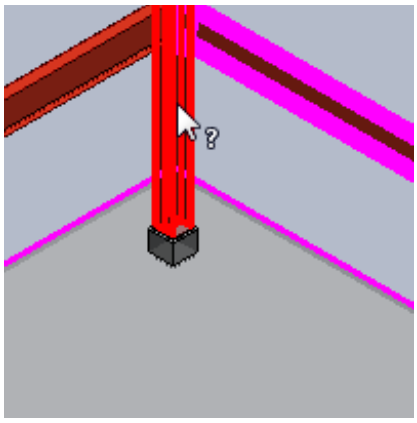
1. Click **Place Stair** on the vertical toolbar.
2. Select the stair to place from the **Select Stair** dialog box. This selection becomes the default selection the next time you place a stair. You can change the default using the **Type** option.
3. Select the top edge for the stair. This edge defines the top elevation of the stair.



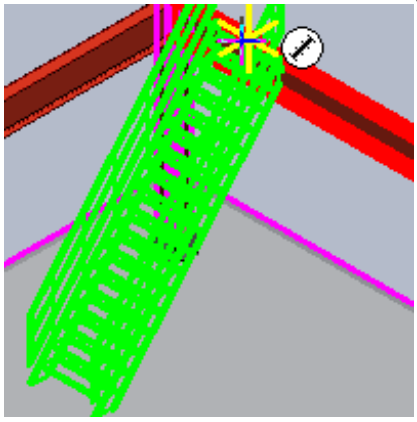
4. Select the bottom plane for the stair. This plane defines the bottom elevation of the stair.



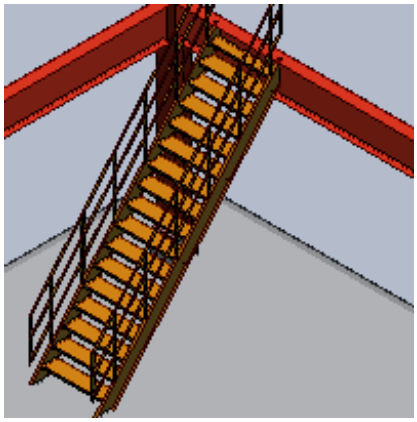
5. Select the optional reference edge for the stair.



6. Define the location of the stair along the top edge.



7. Using the **Horizontal Offset**, **Width**, **Angle**, **Pitch**, **Side**, and **Vertical Offset** options, refine the stair placement.
8. Click **Finish**.



Edit stair placement

1. Click **Select** on the vertical toolbar.
2. Select **Stairs** in the **Locate Filter**.
3. Select the stairs to edit.
4. Using the ribbon bar options, edit the stair placement as needed.

Delete a Stair

1. Click **Select** on the vertical toolbar.
2. Select **Stairs** in the **Locate Filter**.
3. Select the stairs to delete.
4. Press **DELETE**.

Stair Properties Dialog Box

Specifies the properties for the stair that you are editing.

See Also

Definition Tab (Stair Properties Dialog Box) (on page 397)
Occurrence Tab (Stair Properties Dialog Box) (on page 389)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

Occurrence Tab (Stair Properties Dialog Box)

The **Occurrence** tab displays the stair properties that you can edit or that are automatically determined by the software at placement. The property name displays on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one stair, and then selected the properties command, only the common properties between the selected stairs display.

When you view properties for a single stair, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on properties. You can access this document using the **Help > Printable Guides** command in the software.

Category

Select the properties that you want to view for the stair. Stair properties are divided into several different categories: **Standard**, **Weight and CG**, **Fabrication and Construction**, **Surface Treatment and Coating**, and **Responsibility**. You select the category for which you want to define values by using the **Category** option.

Standard

The standard properties that display depend on the reference data for the stair. Because stair reference data is fully customizable, only the common properties are documented.

Name

Displays the name of the stair. The stair name is based on the **Name Rule** selection. If you want to type a new name for the stair, select **User Defined** in the **Name Rule** box, and then type a name for the stair in the **Name** box.

Name Rule

Specify the naming rule to use to name this stair.

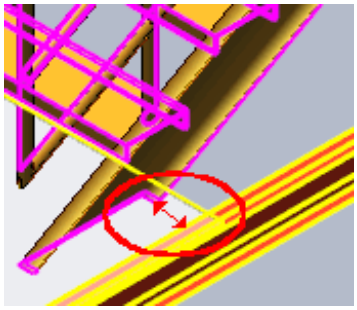
- **Default Name Rule** - Names the stairs using the format "<stair type>-<location>-<index>" where <stair type> is what you select in the **Type** box on the ribbon, <location> is the global workshare location ID and <index> is a unique index number that starts at 0001. For example, StairA1-1-0043.
- **Unique Name Rule** - Names the stairs using the format "<system>-<stair type>-<location>-<index>" where <system> is the name of the parent system to which the stairs belong, <stair type> is what you select in the **Type** box on the ribbon, <location> is the global workshare location ID, and <index> is a unique index number that starts at 0001. For example, Structure System-StairA1-1-0043.
- **User Defined** - Select to specify the stair name yourself in the **Name** box.

System

Select the system to which the stair that you are placing belongs.

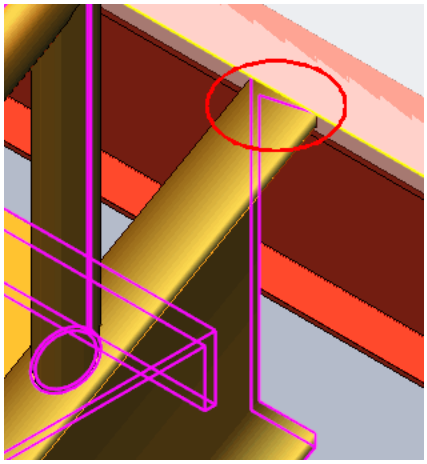
Horizontal Offset

Specify the distance between the outside edge of the stair and the selected reference edge or object.



Vertical Offset

Specify the offset between the top of the stair stringer and the selected top edge.



Side Frame Section Name

Defines the cross-section for the stair stringers.

Handrail Section Name

Defines the cross-section for the stair handrails.

Step Section Name

Defines the cross-section for the stair treads.

Primary Material

Enter the primary material for the stairs.

Primary Grade

Enter the primary material grade for the stairs.

Actual Step Pitch

Displays the actual vertical distance between the stair treads. The software calculates this value using the vertical distance between the top edge and the bottom plane and the stair angle. You can define the maximum allowable pitch between treads in the **Step Pitch** property.

Reporting Requirements

Specify whether this stair is reported.

Reporting Type

Select the reporting requirements code for the stair. Valid codes are defined in the **Catalog** task in the **Reporting Type** select list.

Number of Steps

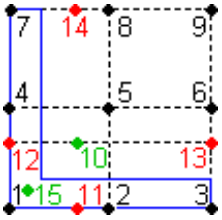
Displays the number of treads on the stair. This value is calculated by the software and cannot be edited.

Justification

Specifies how the stairs align with the connecting object.

Side Frame Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

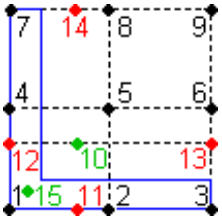


Side Frame Section Angle

Enter the angle, in degrees or radians, by which the side frame cross-section rotates about the member axis. The zero-degree position is either the global Z-axis or the global X-axis, depending on the member orientation.

Handrail Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

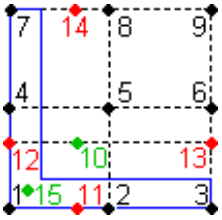


Handrail Section Angle

Enter the angle, in degrees or radians, by which the handrail cross-section rotates about the member axis. The zero-degree position is either the global Z-axis or the global X-axis, depending on the member orientation.

Step Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.



Step Section Angle

Enter the angle, in degrees or radians, by which the step tread's cross-section rotates about the member axis. The zero-degree position is either the global Z-axis or the global X-axis, depending on the member orientation.

With Top Landing

Indicates whether the stair includes a top landing.

Top Landing Length

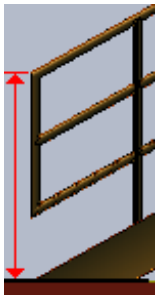
Enter the length dimension of the top landing for the stairs. Smart 3D only uses this dimension if **With Top Landing** is set to **True**.

Platform Thickness

Enter the thickness dimension for the stair platform.

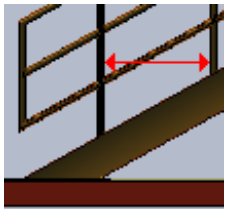
Post Height

Enter the height dimension for the hand rail post.



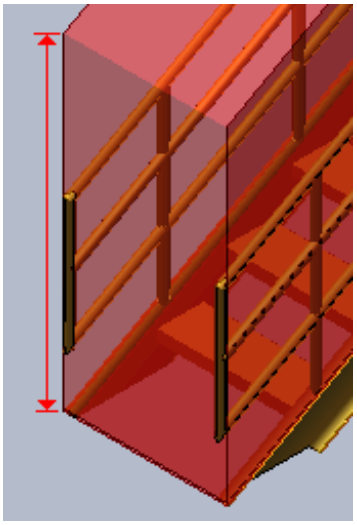
Handrail Post Pitch

Enter the distance between the handrail posts.



Number of Midrails

Enter the number of midrails for the stair handrailing.



Is Assembly

This value is not currently used by the software. Set to **False**.

Envelope Height

Enter the height dimension for the stair operation envelope. This dimension is used by interference checking to check for head clearance above the stair.

Is System

This value is not currently used by the software. Set to **False**.

Span

Displays the length of the stair stringers. This distance is calculated by the software and cannot be edited.

Height

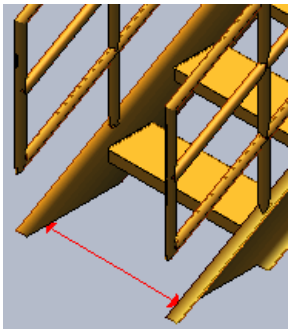
Displays the vertical distance between the select top edge and the bottom plane.

Length

Displays the length of the stair. This distance is calculated by the software and cannot be edited.

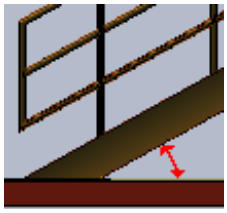
Width

Enter the width of the stair treads. This dimension does not include the width of the stringers and therefore is less than the overall width of the stairs.



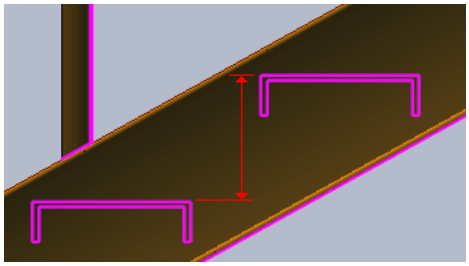
Angle

Enter the angle of inclination between the selected bottom plane and the stair stringers.



Step Pitch

Enter the maximum allowable vertical distance between the stair treads. The actual distance is calculated by the software using the vertical distance between the top edge and the bottom plane and the stair angle. The actual distance is displayed in **Actual Step Pitch**.



Side

Places the stair on the opposite side of the top edge.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material density multiplied by the object's solid volume. Therefore, the material of the object affects the

weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight

Specifies the dry weight of the object.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Dry WCG Origin

Specifies the way in which the dry weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the dry weight center-of-gravity location relative to the active coordinate system.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Surface Treatment and Coating

Exterior Coating Requirement

Specifies the coating requirement for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Exterior Coating Area

Specifies the area of the coating for the object.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

See Also

Stair Properties Dialog Box (on page 389)

Definition Tab (Stair Properties Dialog Box)

The **Definition** tab displays the stair properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one stair, and then selected the properties command, only the common properties between the selected stairs display.

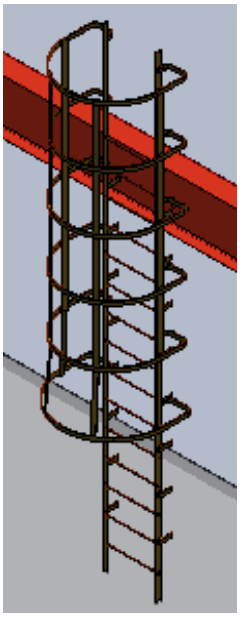
The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties. You can access this document using the **Help > Printable Guides** command in the software.

See Also

Stair Properties Dialog Box (on page 389)

SECTION 26

Place Ladders



Places a ladder in the model.

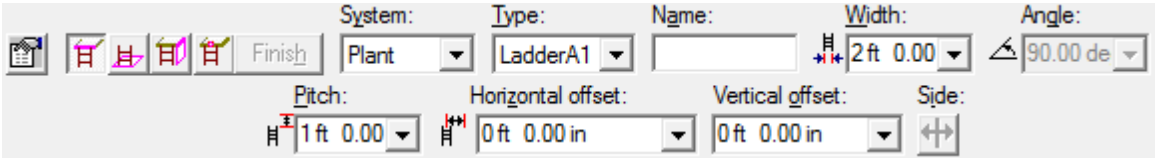
You attach the top of the ladder to an edge of any object. You attach the bottom of the ladder to a plane. The bottom plane must be lower than the top edge. You must also define the position of the ladder along the top edge. There are two methods for defining this position: 1) Select a reference edge or plane that intersects with the top edge and set the horizontal offset distance along the top edge from the intersection point. You can position the ladder on either side of the reference. 2) Define a position for the ladder graphically.

After you define the position of the ladder, the ladder appears in the model. The software automatically calculates the number of rungs.

NOTE Because the ladder, the top edge, and the bottom plane are connected, the interference checker service will not find any hard interference between any part of the ladder (safety cage, rung, and so forth) and the top edge object or the bottom plane.

Place Ladder Ribbon

Specifies the properties for the ladder that you are placing or editing.



Ladder Properties

Activates the **Ladder Properties** dialog box. You can use this dialog box to specify additional ladder properties, such as material and material grade, which you cannot set on the ribbon. For more information, see *Ladder Properties Dialog Box* (on page 404).

Select Top Edge

Select the reference plane or edge that defines the top elevation of the ladder. If this edge is deleted after the ladder has been placed, the software will put the ladder on the To Do List.

Select Bottom Plane

Select the reference plane that defines the bottom elevation of the ladder. This plane must be lower than the top edge that you selected. The reference plane can be flat or sloped up to 45 degrees. If this plane is deleted after the ladder has been placed, the software will put the ladder on the To Do List.

Select Reference Edge

Select a plane or edge from which to place the ladder along the top edge. This input is optional.

Select Position

Graphically define the location of the ladder along the selected top edge.

Finish

Places the ladder in the model.

System

Select the system to which the ladder belongs. You can create new systems in the Systems and Specifications task.

Type

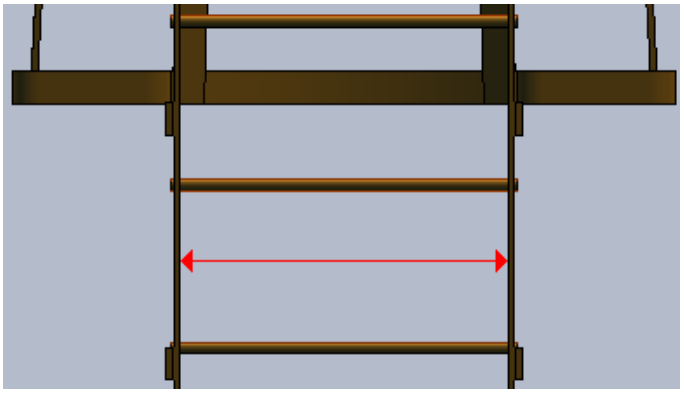
Select the type of ladder to place.

Name

Specify the name for the ladder that you are placing.

Width

Enter the width of the ladder rungs. This dimension does not include the width of the side frames or safety hoop and therefore is less than the overall width of the ladder.

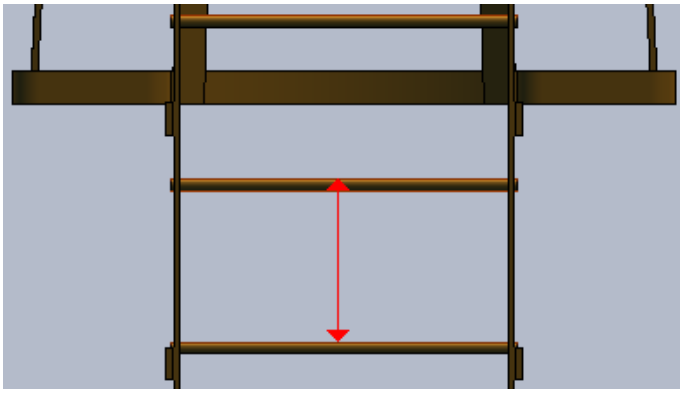


Angle

Enter the angle of inclination between the bottom support and the side frame. This angle is the slope of the ladder. The default angle is 90 degrees.

Pitch

Enter the maximum allowable vertical distance between the ladder rungs.



Horizontal Offset

Specify the distance from the selected reference edge along the specified top edge.

Vertical Offset

Specify the offset between the top of the ladder and the selected top edge. This distance is in addition to the define **Top Extension** distance.

Side

Places the ladder on the opposite side of the top edge.

Select Ladder Dialog Box

Allows selection of the type of ladders to be placed. This dialog box appears when you select **More** in the **Type** list. By browsing through the hierarchy, you can find any ladder in the Catalog database. After you select a ladder, the software returns you to the model where you can finalize placement.

 **Back**

Returns you to the previously selected ladder type or node. Use this command to navigate through the hierarchy to the specific type that you need.

 **Forward**

Sends you to the last selected ladder type or node that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific type that you need.

 **Up One Level**

Brings up the next highest level of the catalog hierarchy. Use this command to navigate through the hierarchy to the specific type that you need.

 **Properties**

Displays the properties of the selected ladder. Because you cannot modify any properties until the ladder is placed, all properties on the dialog box are read-only.

 **Preview**

Displays a picture of the selected ladder. The image file must be assigned to the ladder in the reference data.

 **List View**

Sets the dialog box to display ladders in a list view.

 **Grid View**

Sets the dialog box to display ladders in a spreadsheet-style grid view.

Address

Specifies your exact location within the displayed hierarchy.

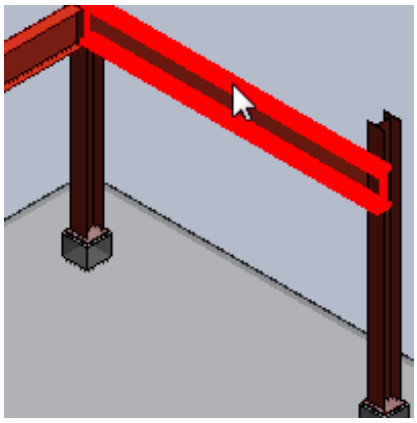
What do you want to do?

- *Place a ladder* (on page 401)

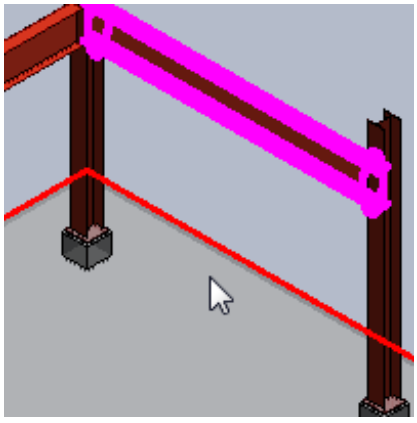
- *Edit ladder placement* (on page 403)
- *Edit ladder properties* (on page 403)
- *Delete a ladder* (on page 403)

Place a ladder

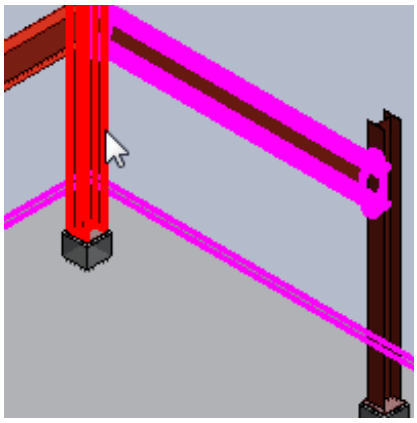
1. Click **Place Ladder** on the vertical toolbar.
2. Select the ladder to place from the **Select Ladder** dialog. This selection becomes the default selection the next time that you place a ladder. You can change the default using the **Type** option.
3. Select the top edge for the ladder. This edge defines the top elevation of the ladder.



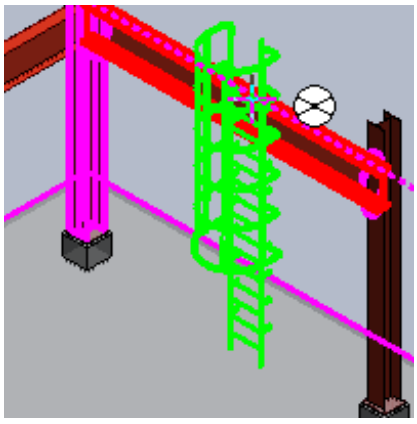
4. Select the bottom plane for the ladder. This plane defines the bottom elevation of the ladder.



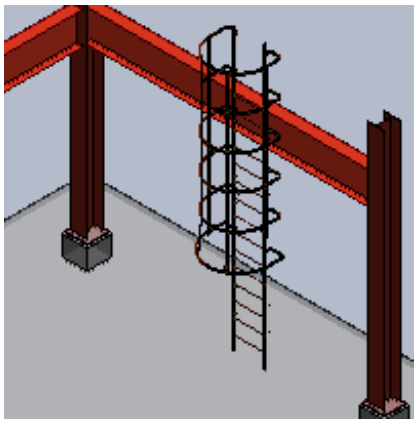
5. Select the optional reference edge for the ladder.



6. Define the location of the ladder along the top edge.



7. Using the **Horizontal Offset**, **Width**, **Angle**, **Pitch**, **Side**, and **Vertical Offset** options, refine the ladder placement.
8. Click **Finish**.



Edit ladder placement

1. Click **Select** on the vertical toolbar.
2. Select **Ladders** in the **Locate Filter**.
3. Select the ladder to edit.
4. Using the ribbon bar options, edit the ladder position as needed.

Edit ladder properties

1. Click **Select** on the vertical toolbar.
2. Select **Ladders** in the **Locate Filter**.
3. Select the ladder to edit.
4. Click **Edit > Properties**.
5. Edit the properties as needed. For more information about the properties, see *Ladder Properties Dialog Box* (on page 404).

Delete a ladder

1. Click **Select** on the vertical toolbar.
2. Select **Ladders** in the **Locate Filter**.
3. Select the ladder to delete.
4. Click **Delete**.

Ladder Properties Dialog Box

Specifies the properties for the ladder that you are editing.

See Also

Definition Tab (Ladder Properties Dialog Box) (on page 417)
Occurrence Tab (Ladder Properties Dialog Box) (on page 404)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)
Place Ladders (on page 398)

Occurrence Tab (Ladder Properties Dialog Box)

The **Occurrence** tab displays the ladder properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one ladder, and then selected the properties command, only the common properties between the selected ladders display.

When viewing properties for a single ladder, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on properties. You can access this document using the **Help > Printable Guides** command in the software.

Category

Select the properties that you want to view for the ladder. Ladder properties are divided into several different categories: **Standard**, **Weight and CG**, **Fabrication and Construction**, **Surface Treatment and Coating**, and **Responsibility**. You select the category to define respective values by using the **Category** option.

Standard

Standard properties that are displayed depend on the reference data of the ladder. Because ladder reference data is fully customizable, only the common properties are documented.

Name

Displays the name of the ladder. The ladder name is based on the **Name Rule** selection. If you want to type a new name for the ladder, in the **Name Rule** box, select **User Defined**, and then type a name for the ladder in the **Name** box.

Name Rule

Specify the naming rule that you want to use to name this ladder.

- **Default Name Rule** - Names the ladder using the format "<ladder type>-<location>-<index>" where <ladder type> is what you select in the **Type** box on the ribbon, <location> is the global workshare location ID, and <index> is a unique index number that starts at 0001. For example, LadderA1-1-0043.
- **Unique Name Rule** - Names the ladder using the format "<system>-<ladder type>-<location>-<index>" where <system> is the name of the parent system to which the ladder belong, <ladder type> is what you select in the **Type** box on the ribbon, <location> is the global workshare location ID, and <index> is a unique index number that starts at 0001. For example, Structure System-LadderA1-1-0043.

- **User Defined** - Select to specify the ladder name yourself in the **Name** box.

System

Select the system to which the ladder that you are placing belongs.

Horizontal Offset

Specify the distance from the selected reference edge along the specified top edge.

Vertical Offset

Specify the offset between the top of the ladder and the selected top edge. This distance is in addition to the define **Top Extension** distance.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade. To change the options on the list, edit the **Structure > Materials > Properties** node in Catalog.

Reporting Requirements

Specify whether this object is reported.

Reporting Type

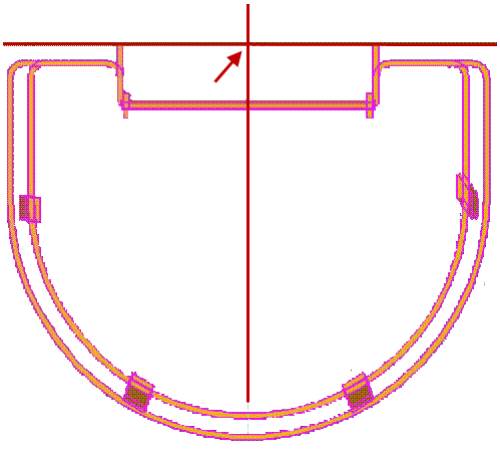
Select the reporting requirements code for the object. Valid codes are defined in Catalog in the **Reporting Type** select list.

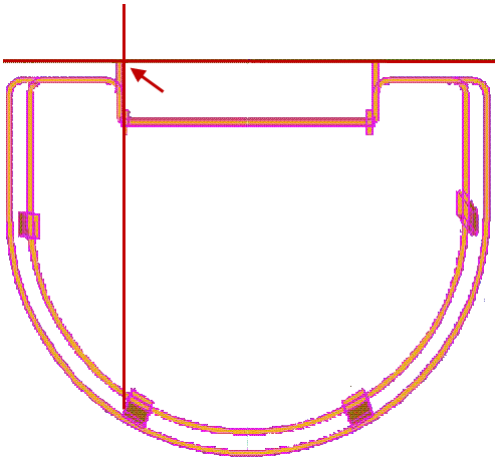
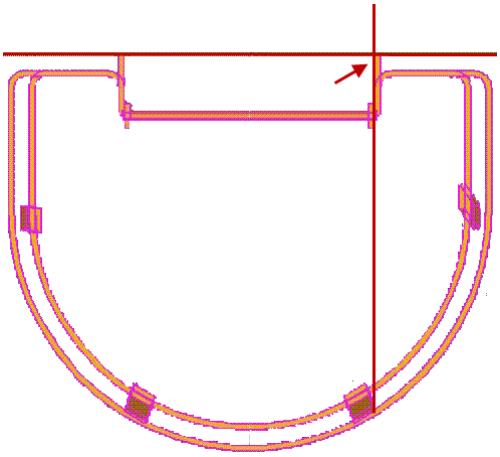
With Wall Supports

Indicates whether the ladder includes wall supports.

Justification

Specifies how the object aligns with the connecting structural objects.

	Center

	Left
	Right

First Step Pitch

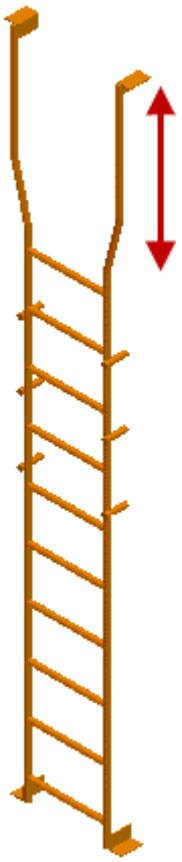
Displays the distance between the bottom reference plane and the first rung on the ladder. This distance is calculated by the software and cannot be edited.

Number of Steps

Displays the number of rungs on the ladder. This number is calculated by the software and cannot be edited.

Top Extension

Enter the distance the ladder extends above the selected top edge.



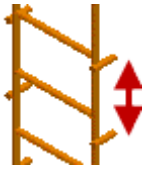
Bottom Extension

Enter the distance between the selected bottom plane and the start, or bottom, of the ladder. This distance must be a positive number. You cannot extend a ladder below the bottom plane using this property.



Support Leg Pitch

Enter the vertical distance, along the ladder, between the wall supports.



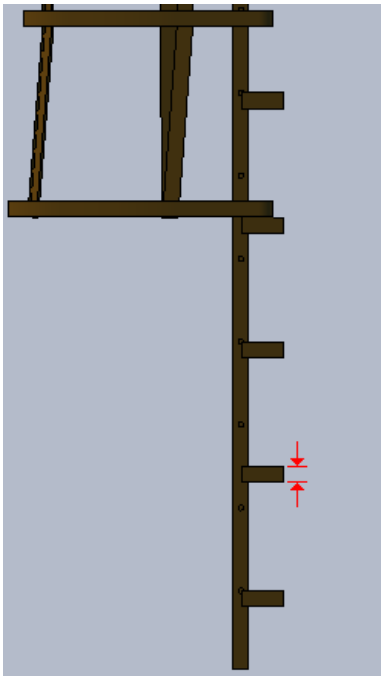
Support Leg Thickness

Enter the thickness of the wall support legs. This distance is left-right distance of the support leg if you were standing facing the ladder.



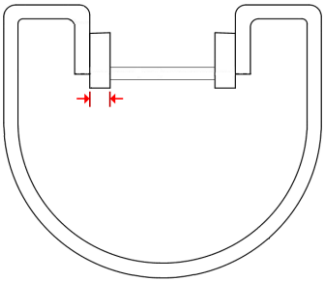
Support Leg Width

Enter the width of the wall support legs.



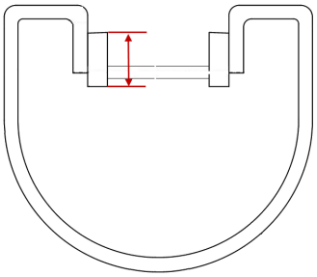
Side Frame Thickness

Enter the thickness dimension for the side frame.



Side Frame Width

Enter the width dimension for the side frame.

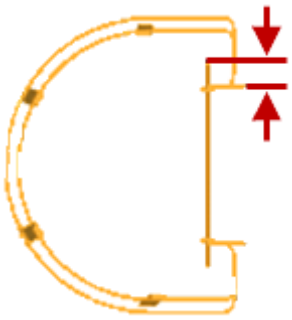


Step Diameter

Enter the diameter dimension for the step or rung.

Step Protrusion

Enter the dimension for the rung protrusion. This dimension is the distance the ladder rungs extend past the outside side frames.



VlDim1

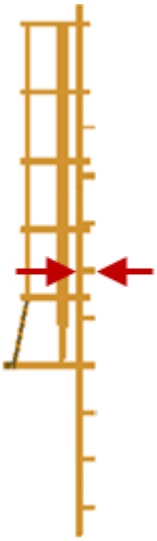
This value is not currently used by the SPSLadderMacros.LadderTypeA symbol.

VlDim2

This value is not currently used by the SPSLadderMacros.LadderTypeA symbol.

VlDim3

This value is not currently used by the SPSLadderMacros.LadderTypeA symbol.



Wall Offset

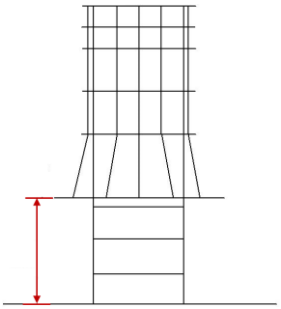
Enter the dimension by which the ladder is offset from the supporting wall.

With Safety Hoop

Indicates whether the ladder includes a safety hoop.

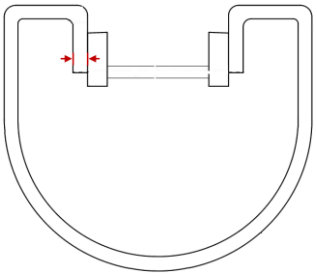
Bottom Hoop Level

Enter the dimension for the distance between the bottom of the hoop and the bottom of the ladder.



Hoop Plate Thickness

Enter the thickness dimension for the hoop plate.

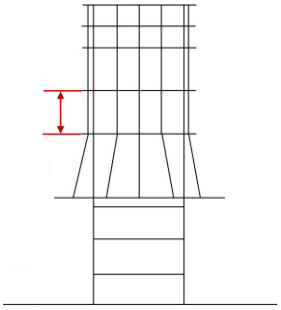


Hoop Plate Width

Enter the dimension for the hoop plate width.

Hoop Pitch

Enter the distance between the hoops.

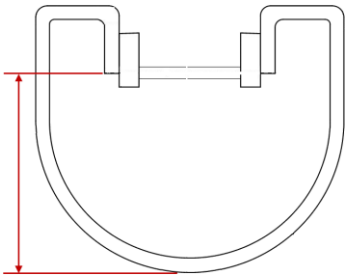


Max Hoop Pitch

Enter the maximum distance between the hoops.

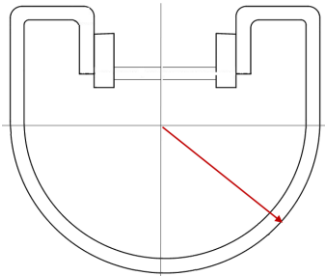
Hoop Clearance

Enter the dimension for the hoop clearance.



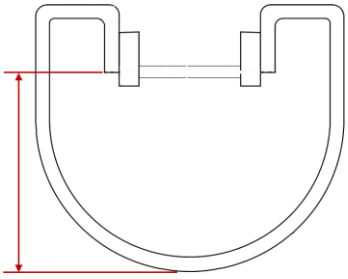
Hoop Radius

Enter the radius dimension for the safety hoop.



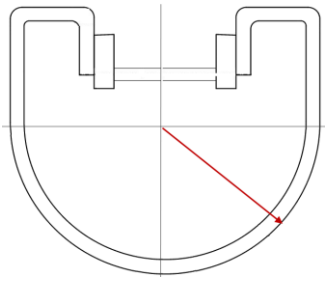
Flare Clearance

Enter the clearance dimension for the hoop flare.



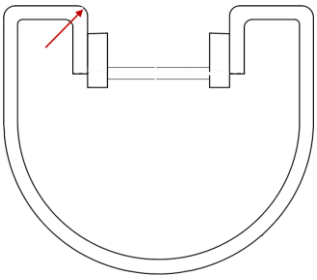
Flare Radius

Enter the radius dimension for the hoop flare.



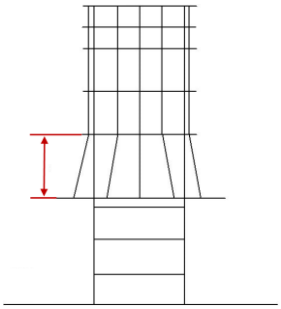
Hoop Flare Bend Radius

Enter the radius dimension for the hoop flare bend.



Hoop Flare Height

Enter the height dimension for the hoop flare.



Vertical Strap Count

Enter the number of vertical straps.

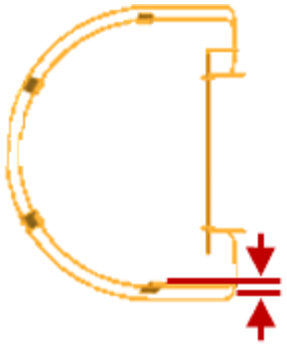
Vertical Strap Width

Enter the width dimension for the vertical strap.



Vertical Strap Thickness

Enter the thickness dimension for the vertical strap.



Hoop Opening

Enter the number associated with the hoop opening type.

Is Assembly

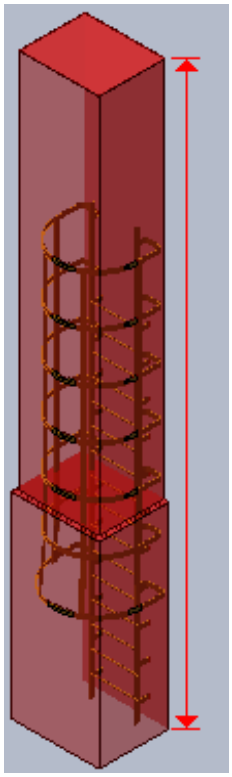
This value is not currently used by the software. Set to **0**.

Is System

This value is not currently used by the software. Set to **0**.

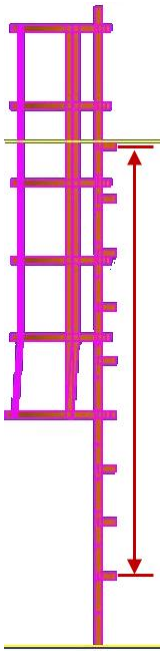
Envelope Height

Enter the operation envelope height for the ladder. This dimension is used by interference checking.



Span

Enter the distance from the top support to the bottom support. This is the default distance. When placing the ladder, the ISPSStairSymbolServices component calculates the actual distance.



Height

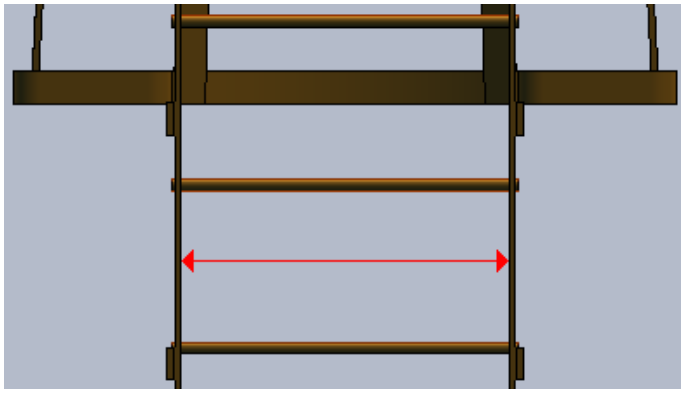
Displays the distance between the top edge and the bottom reference plane. This value is calculated by the software.

Length

Displays the length of the ladder. This value is calculated by the software.

Width

Enter the width of the ladder rungs. This dimension does not include the width of the side frames or safety hoop and therefore is less than the overall width of the ladder.

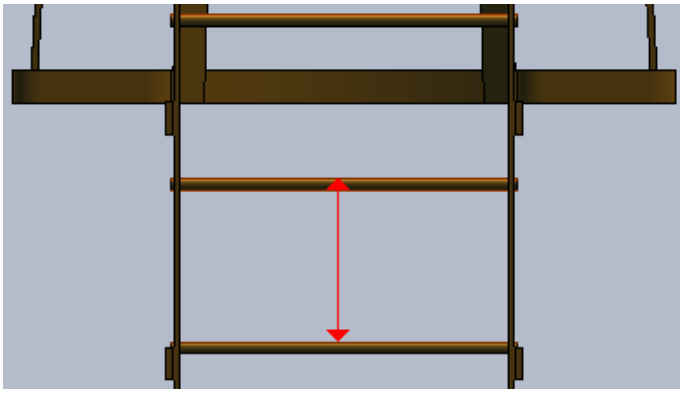


Angle

Enter the angle of inclination between the bottom support and the side frame. This angle is the slope of the ladder.

Step Pitch

Enter the maximum allowable vertical distance between the ladder rungs.



Side

Places the ladder on the opposite side of the top edge.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material density multiplied by the object's solid volume. Therefore, the material of the object affects the weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight

Specifies the dry weight of the object.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Dry WCG Origin

Specifies the way in which the dry weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the dry weight center-of-gravity location relative to the active coordinate system.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Surface Treatment and Coating

Exterior Coating Requirement

Specifies the coating requirement for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Exterior Coating Area

Specifies the area of the coating for the object.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list,

edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

See Also

Ladder Properties Dialog Box (on page 404)

Definition Tab (Ladder Properties Dialog Box)

The **Definition** tab displays the ladder properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one ladder, and then selected the properties command, only the common properties between the selected ladders display.

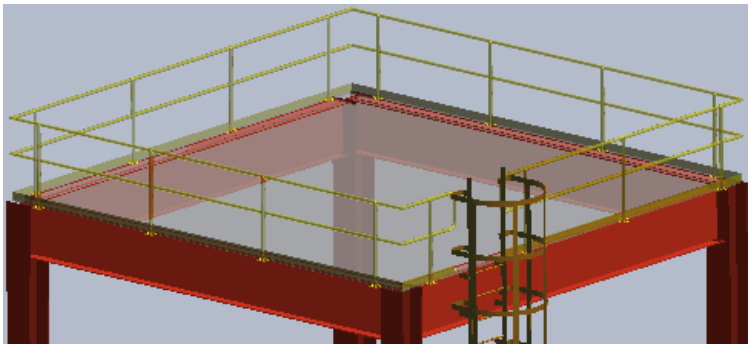
The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties. You can access this document using the **Help > Printable Guides** command in the software.

See Also

Ladder Properties Dialog Box (on page 404)

SECTION 27

Place Handrails



Places a handrail along a path that you specify in the model. The handrail path can be straight, curved, or a combination of both and can be on the same plane or transverse multiple elevations.

NOTE Because handrails are placed along a defined path, moving other objects in the model might not affect the position of the handrail. Handrails will move when other objects in the model move if you have defined the handrail path using the Point on Surface or Point on Curve constraints. Use the **Tools > Options > SmartSketch** command to make these two constraints available for handrail path placement.

Place Handrail Ribbon

Specifies the properties for the handrail that you are placing or editing.



Handrail Properties

Activates the **Handrail Properties** dialog box. You can use this dialog box to specify additional handrail properties, such as rail section and toe plate section, which you cannot set on the ribbon. For more information, see *Handrail Properties Dialog Box* (on page 422).

Create Handrail Path

Activates the Create Path Ribbon with which you define the handrail path.

Toggle Side

Select to place the handrail on the other side of the path.

Convert

Decomposes the handrail symbol to individual structural members for each post and rail. You should convert a handrail when you want to add, edit, or delete the positions of individual posts, rails, or perform some other handrail customization that cannot be done using the handrail symbol properties. The Handrail object will become the system parent of the individual members. Deleting the Handrail object will delete the children members. Handrails that you have converted to member parts cannot be converted back to a handrail symbol. You can place handrails as individual members during initial placement by clicking **Convert** instead of **Finish**, or you can select an existing handrail and convert it to member parts.

Finish

Places the handrail in the model.

System

Select the system to which the handrail belongs. You can create new systems in the Systems and Specifications task.

Type

Select the type of handrail to place from the catalog.

Name

Specify the name for the handrail that you are placing. This box is read-only unless you are using the **User Defined** naming rule.

Beginning Treatment

Specify the end treatment for the beginning of the handrail.

End Treatment

Specify the end treatment for the end of the handrail.

Select Handrail Dialog Box

Allows selection of the type of handrails to be placed. This dialog box appears when you select **More** in the **Type** list. By browsing through the hierarchy, you can find any handrail in the Catalog database. After you select a handrail, the software returns you to the model, where you can finalize placement.

 **Back**

Returns you to the previously selected handrail type or node. Use this command to navigate through the hierarchy to the specific type that you need.

 **Forward**

Sends you to the last selected handrail type or node that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific type that you need.

 **Up One Level**

Brings up the next highest level of the catalog hierarchy. Use this command to navigate through the hierarchy to the specific type that you need.

 **Properties**

Displays the properties of the selected handrail. Because you cannot modify any properties until the handrail is placed, all properties on the dialog box are read-only.

 **Preview**

Displays a picture of the selected handrail. The image file must be assigned to the handrail in the reference data.

 **List View**

Sets the dialog box to display handrails in a list view.

Grid View

Sets the dialog box to display handrails in a spreadsheet-style grid view.

Address

Specifies your exact location within the displayed hierarchy.

What do you want to do?

- *Place a handrail* (on page 420)
- *Place a handrail as individual members* (on page 420)
- *Split a handrail* (on page 421)
- *Edit handrail placement* (on page 421)
- *Edit handrail properties* (on page 421)
- *Convert handrail to individual members* (on page 421)
- *Delete a handrail* (on page 422)

Place a handrail

1. Click **Place Handrail** on the vertical toolbar.
2. Select the handrail to place from the **Select Handrail** dialog. This selection becomes the default selection the next time you place a handrail. You can change the default using the **Type** option.
3. Define the handrail path.

   Define a Path (on page 501)
4. Select a system for the handrail.
5. Click **Finish**.

Place a handrail as individual members

1. Click **Place Handrail** on the vertical toolbar.
2. Select the handrail to place from the **Select Handrail** dialog. This selection becomes the default selection the next time you place a handrail. You can change the default using the **Type** option.
3. Define the handrail path.

   Define a Path (on page 501)
4. Select a system for the handrail.
5. Click **Convert**.

Split a handrail

1. Select the handrail to split.
2. Click **Define Handrail Path** .
3. Click **Edit** on the ribbon.
4. Select the handrail path segment to split.
5. Click **Insert Vertex** on the ribbon.
6. Click along the handrail path at the split location to insert the new vertex.
7. Click **Create** on the ribbon, and then click **Edit** again.
8. Select the vertex (the small yellow box) that you just inserted at the split location.
9. On the ribbon for **Turn Type**, select **Split**.
10. Click **Finish**.
11. Click **Finish**.

Edit handrail placement

1. Click **Select** on the vertical toolbar.
2. Select **Handrails** in the **Locate Filter**.
3. Select the handrail to edit.
4. Using the ribbon option, edit the handrail placement as needed.

Edit handrail properties

1. Click **Select** on the vertical toolbar.
2. Select **Handrails** in the **Locate Filter**.
3. Select the handrail to edit.
4. Click **Edit > Properties**.
5. Edit the properties as needed. For more information about these properties, see *Handrail Properties Dialog Box* (on page 422).

Convert handrail to individual members

1. Click **Select** on the vertical toolbar.
2. Select **Handrails** in the **Locate Filter**.
3. Select the handrail to convert.
4. Click **Convert** on the handrail ribbon.

NOTE You should convert a handrail when you want to edit the positions of individual posts, rails, or perform some other handrail customization that cannot be done using the handrail symbol properties. Handrails that you have converted to member parts cannot be converted back to a handrail symbol. To delete a converted handrail, you will need to delete each handrail part individually.

Delete a handrail

1. Click **Select** on the vertical toolbar.
2. Select **Handrails** in the **Locate Filter**.
3. Select the handrail to delete.
4. Press **Delete**.

Handrail Properties Dialog Box

Specifies the properties for the handrail that you are editing.

See Also

Definition Tab (Handrail Properties Dialog Box) (on page 457)
Occurrence Tab (Handrail Properties Dialog Box) (on page 422)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

Occurrence Tab (Handrail Properties Dialog Box)

The **Occurrence** tab displays in a grid the handrail properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one handrail, and then selected the properties command, only the common properties between the selected handrails display.

When viewing properties for a single handrail, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on properties. You can access this document using the **Help > Printable Guides** command in the software.

Category

Select the properties that you want to view for the handrail. Handrail properties are divided into several different categories: **Standard**, **Weight and CG**, **Fabrication and Construction**, **Surface Treatment and Coating**, and **Responsibility**. You select the category to define respective values by using the **Category** option.

Standard

The standard properties that display depend on the reference data for the handrail. Because handrail reference data is fully customizable, only the common properties are documented.

Name

Displays the name of the handrail based on the **Name Rule** selection. If you want to type a

new name for the handrail, select **User Defined**, and then type a name for the handrail in the **Name** box.

Name Rule

Specify the naming rule that you want to use to name this handrail.

- **Default Name Rule** - Names the handrail using the format "<handrail type>-<location>-<index>" where <handrail type> is what you select in the **Type** box on the ribbon, <location> is the global workshare location ID, and <index> is a unique index number that starts at 0001. For example, TMTHandrail-1-0043.
- **Unique Name Rule** - Names the handrail using the format "<system>-<handrail type>-<location>-<index>" where <system> is the name of the parent system to which the handrail belongs, <handrail type> is what you select in the **Type** box on the ribbon, <location> is the global workshare location ID, and <index> is a unique index number that starts at 0001. For example, Structure System-TMTHandrailA1-1-0043.
- **User Defined** - Select to specify the handrail name yourself in the **Name** box.

System

Select the system to which the handrail that you are placing belongs. You can create new systems in the **Systems and Specifications** task.

Top Rail Section Name

Enter the cross-section name for the top rail.

Top Rail Section Reference Standard

Enter the section library name that contains the top rail section.

Mid Rail Section Name

Enter the cross-section name for the mid rail.

Mid Rail Section Reference Standard

Enter the section library name that contains the mid rail section.

Toe Plate Section Name

Enter the cross-section name for the toe plate.

Toe Plate Section Reference Standard

Enter the section library name that contains the toe plate section.

Post Section Name

Enter the cross-section name for the post.

Post Section Reference Standard

Enter the section library name that contains the post's section.

Primary Material

Specify the material for the handrail sections.

Primary Grade

Specify the material grade for the handrail sections.

Reporting Requirements

Specify whether this handrail is reported.

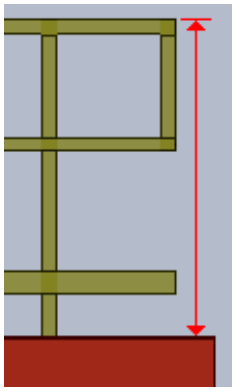
Reporting Type

Select the reporting requirements code for the handrail. Valid codes are defined in the **Catalog** task in the **Reporting Type** select list.

Type A Handrail (TMTHandrail and TMHandrail)

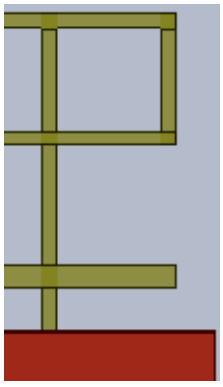
Height

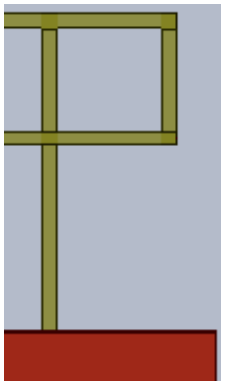
Enter the height of the handrail from the path to the top of the top rail.



With Toe Plate

Select **True** to attach a toe plate to the handrail. Select **False** to place a handrail without a toe plate.





Number of Midrails

Enter the number of mid rails for the handrail. The first mid rail is placed using the defined **To Top of Mid Rail Distance**. The subsequent mid rails are placed above that using the **Mid Rail Spacing** distance.

Horizontal Offset Distance

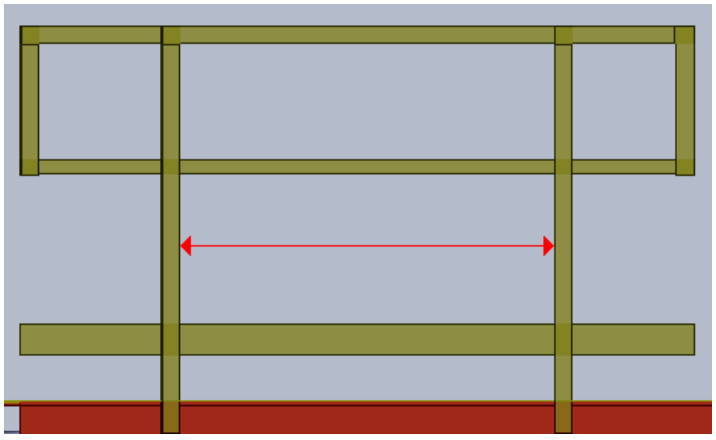
Enter the horizontal offset from the defined handrail path to the handrail. You control the direction of the offset using the **Horizontal Offset** property.

Handrail Orientation

Defines the orientation of the post when the handrail is on a slope. Select **Always Vertical** to keep the posts vertical even when the handrail is on a slope. Select **Perpendicular to Slope** to keep the posts perpendicular to the slope.

Segment Maximum Spacing

Enter the maximum distance allowed between the posts.

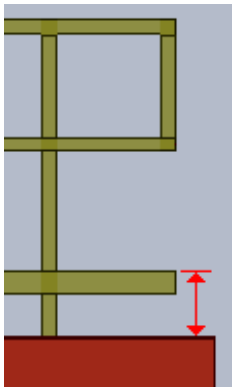


Sloped Segment Maximum Spacing

Enter the maximum distance allowed between posts for that part of a handrail that is on a slope.

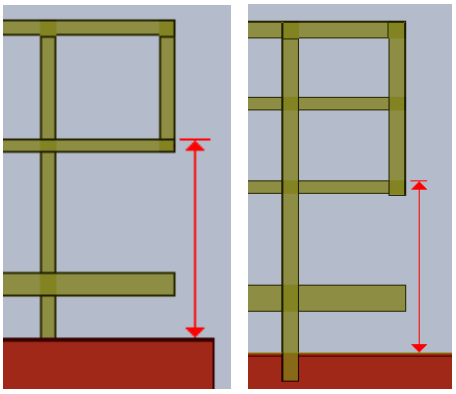
Distance to Top of Toe Plate

Specify the height of the toe plate measured from the path.



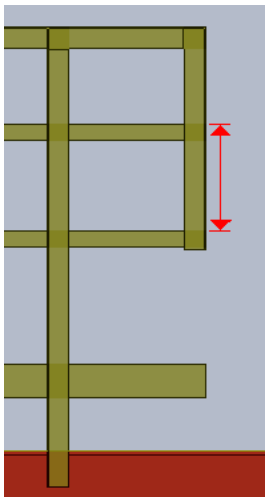
To Top of Midrail Distance

Specify the distance between the path and the top of the mid rail. If you are placing more than one mid rail, this distance is to the top of the bottommost mid rail.



Midrail Spacing

Specify the distance between the mid rails if you are placing more than one mid rail. This value is ignored if you have only one mid rail.

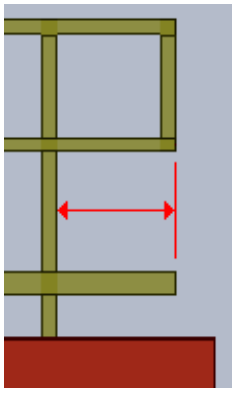


With Post At Turn

Specifies if you want a post in the handrail corner. Select **True** to place a post in the corner. Select **False** to not require a post in the corner.

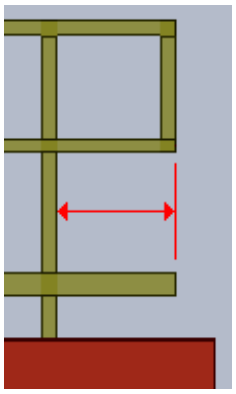
Begin Extension Length

Enter the distance between the beginning of the handrail and the first handrail post. This distance must be greater than zero.



End Extension Length

Enter the distance between the end of the handrail and the last handrail post. This distance must be greater than zero.



Is Assembly

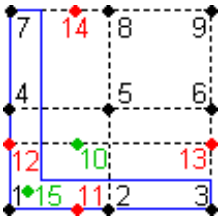
Not used by the software. Set to **False**.

Is System

Not used by the software. Set to **False**.

Top Rail Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

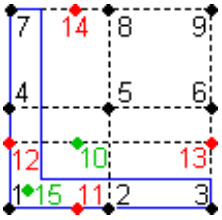


Top Rail Section Angle

Defines the angle, in degrees or radians, by which the top rail cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis depending on the member orientation.

Midrail Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

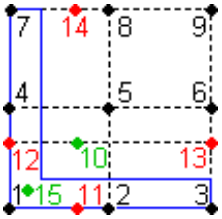


Midrail Section Angle

Defines the angle, in degrees or radians, by which the mid rail cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis, depending on the member orientation.

Toe Plate Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

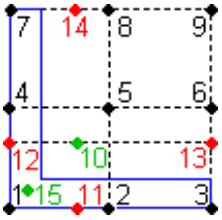


Toe Plate Section Angle

Defines the angle, in degrees or radians, by which the toe plate cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis depending on the member orientation.

Post Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.



Post Section Angle

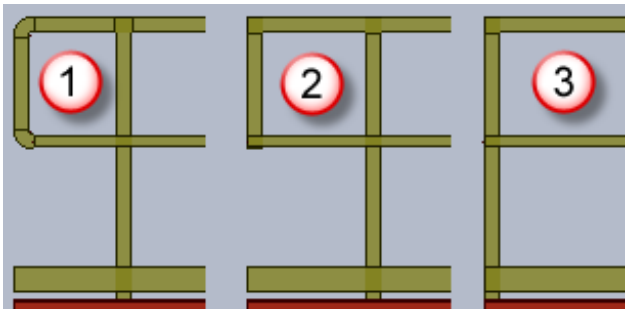
Defines the angle, in degrees or radians, by which the post cross-section is rotated about the member axis. The zero degree position is either the global Z-axis or the global X-axis, depending on the member orientation.

Horizontal Offset

Select the direction the **Horizontal Offset Distance** property value is applied.

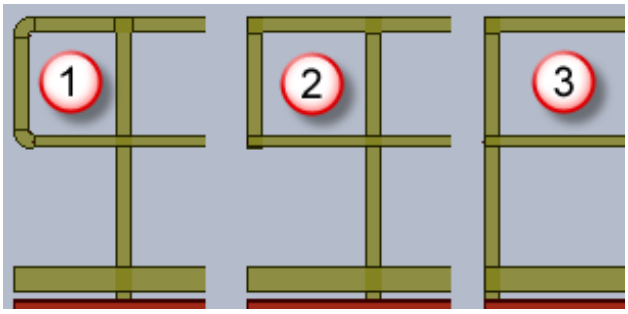
Begin Treatment

Specify a circular (1), rectangular (2), or none (3) end treatment for the beginning of the handrail.



End Treatment

Specify a circular (1), rectangular (2), or none (3) end treatment for the end of the handrail.



Post Connection Type

Select how you want the handrail posts connected to the supporting structure.

Total Length

Displays the length of the handrail path.

Type A Handrail (SideMountedtoMemberMTHandrail)

Connection Type for Side Mount

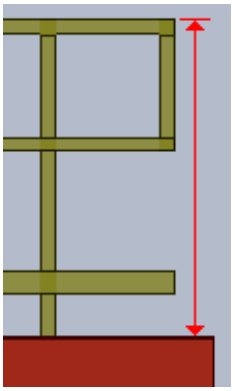
Specifies the type of side mount for the handrail. You can select a side mount with a pad, or a side mount with a bracket.

Vertical offset value for Side Mount

Specify the vertical offset between the bottom of the posts and the bottom of the toe plate.

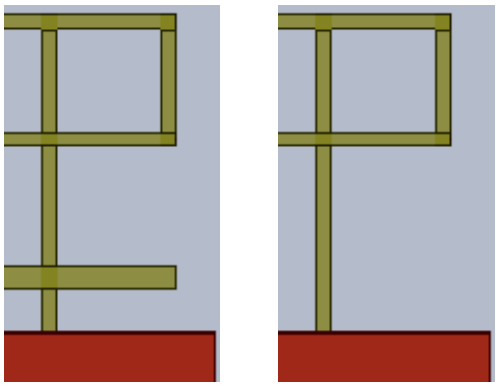
Height

Enter the height of the handrail from the path to the top of the top rail.



With Toe Plate

Select **True** to attach a toe plate to the handrail. Select **False** to place a handrail without a toe plate.



Number of Midrails

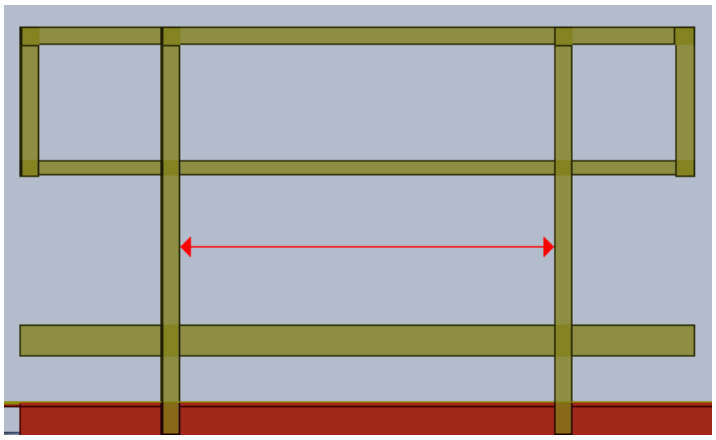
Enter the number of mid rails for the handrail. The first mid rail is placed using the defined **To Top of Mid Rail Distance**. The subsequent mid rails are placed above that using the **Mid Rail Spacing** distance.

Handrail Orientation

Defines the orientation of the post when the handrail is on a slope. Select **Always Vertical** to keep the posts vertical even when the handrail is on a slope. Select **Perpendicular to Slope** to keep the posts perpendicular to the slope.

Segment Maximum Spacing

Enter the maximum distance allowed between the posts.

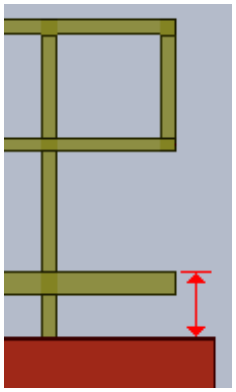


Sloped Segment Maximum Spacing

Enter the maximum distance allowed between posts for that part of a handrail that is on a slope.

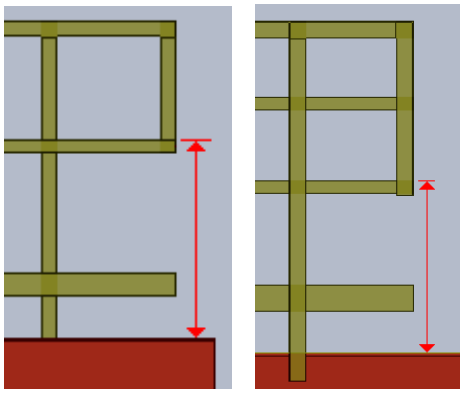
Distance to Top of Toe Plate

Specify the height of the toe plate measured from the path.



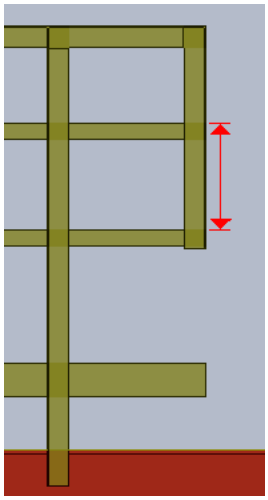
To Top of Midrail Distance

Specify the distance between the path and the top of the mid rail. If you are placing more than one mid rail, this distance is to the top of the bottommost mid rail.



Midrail Spacing

Specify the distance between the mid rails if you are placing more than one mid rail. This value is ignored if you have only one mid rail.



With Post At Turn

Specifies if you want a post in the handrail corner. Select **True** to place a post in the corner. Select **False** to not require a post in the corner.

Is Assembly

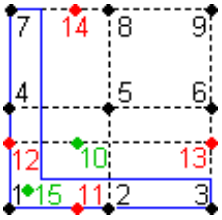
Not used by the software. Set to **False**.

Is System

Not used by the software. Set to **False**.

Top Rail Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

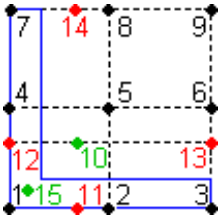


Top Rail Section Angle

Defines the angle, in degrees or radians, by which the top rail cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis depending on the member orientation.

Midrail Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

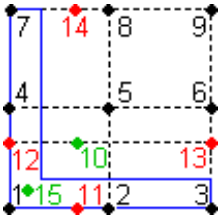


Midrail Section Angle

Defines the angle, in degrees or radians, by which the mid rail cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis, depending on the member orientation.

Toe Plate Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.



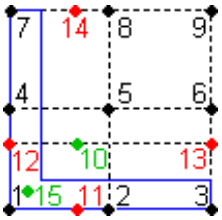
Toe Plate Section Angle

Defines the angle, in degrees or radians, by which the toe plate cross-section is rotated

about the member axis. The zero-degree position is either the global Z-axis or the global X-axis depending on the member orientation.

Post Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.



Post Section Angle

Defines the angle, in degrees or radians, by which the post cross-section is rotated about the member axis. The zero degree position is either the global Z-axis or the global X-axis, depending on the member orientation.

Horizontal Offset

Select the direction the **Horizontal Offset Distance** property value is applied.

Horizontal Offset Distance

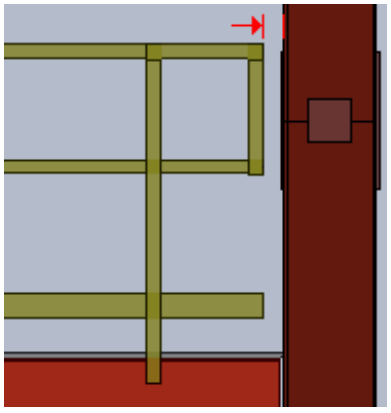
Enter the horizontal offset from the defined handrail path to the handrail. You control the direction of the offset using the **Horizontal Offset** property.

Floor Thickness

Enter the thickness of the floor. The thickness is used to calculate the correct height for the toe plate.

Column Clearance

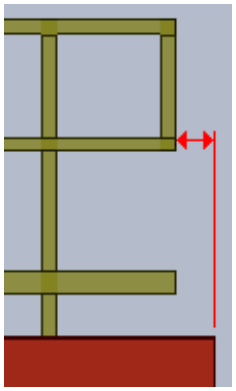
Enter the distance between the column, if one exists, and the handrail. If a column does not exist, this value is ignored and the **End Clearance** value is used instead.



End Clearance

Enter the distance between the member end and the handrail end. This value is only used

when a column does not exist at that member end.



Mirror

Flips the handrail to the other side of the path or member.

Start Column Offset

Enter the offset distance between the column at the start end of the handrail and the handrail. This offset is in addition to the **Column Clearance** offset. Use this option when you want an offset only at the start of the handrail. This option has no effect if there is no column at the start end of the member on which the handrail is placed.

End Column Offset

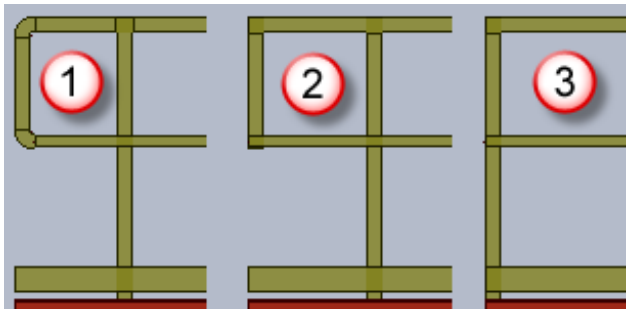
Enter the offset distance between the column at the end of the handrail and the handrail. This offset is in addition to the **Column Clearance** offset. Use this option when you want an offset only at the end of the handrail. This option has no effect if there is no column at the end of the member on which the handrail is placed.

Offset Reference

Select from where on the member you want to measure the handrail horizontal offset. You can select to use the inner port-face, the centerline (axis), or the outer port-face of a bounding member.

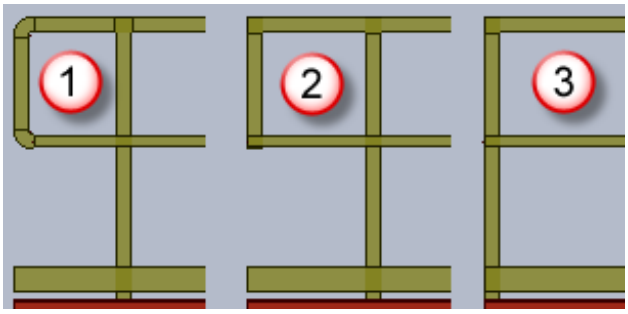
Begin Treatment

Specify a circular (1), rectangular (2), or none (3) end treatment for the beginning of the handrail.



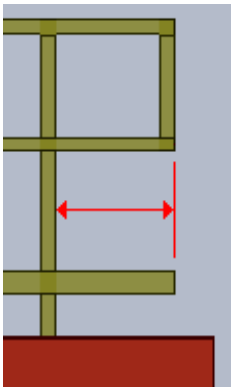
End Treatment

Specify a circular (1), rectangular (2), or none (3) end treatment for the end of the handrail.



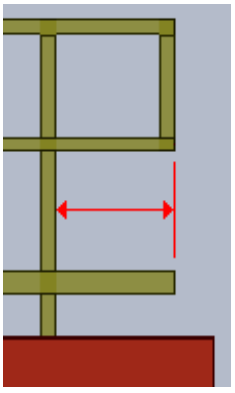
Begin Extension Length

Enter the distance between the beginning of the handrail and the first handrail post. This distance must be greater than zero.



End Extension Length

Enter the distance between the end of the handrail and the last handrail post. This distance must be greater than zero.



Minimum offset Clearance at Post Turn

Enter the minimum distance from a handrail turn that the software can place a post.

Maximum offset Clearance at Post Turn

Enter the maximum distance from a handrail turn that the software can place a post.

Length

Displays the total length of the handrail.

Type A Handrail (SideMounted)

Connection Type for Side Mount

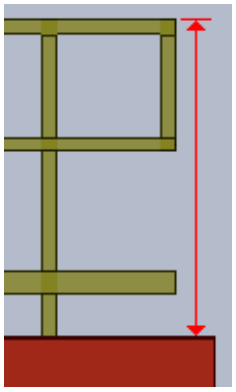
Specifies the type of side mount for the handrail. You can select a side mount with a pad, or a side mount with a bracket.

Vertical offset value for Side Mount

Specify the vertical offset between the bottom of the posts and the bottom of the toe plate.

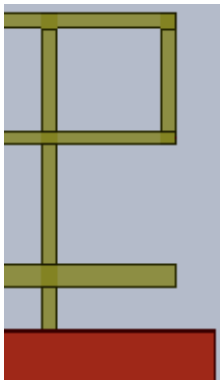
Height

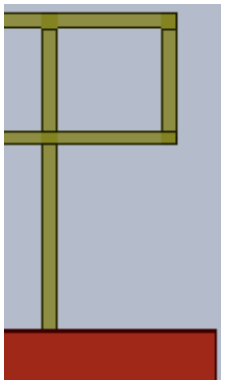
Enter the height of the handrail from the path to the top of the top rail.



With Toe Plate

Select **True** to attach a toe plate to the handrail. Select **False** to place a handrail without a toe plate.





Number of Midrails

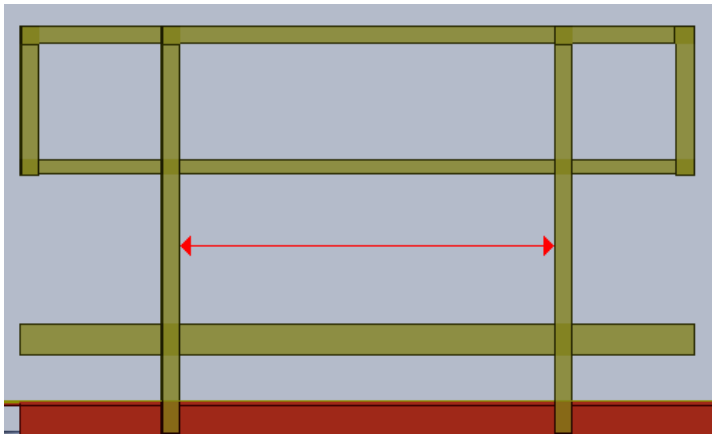
Enter the number of mid rails for the handrail. The first mid rail is placed using the defined **To Top of Mid Rail Distance**. The subsequent mid rails are placed above that using the **Mid Rail Spacing** distance.

Handrail Orientation

Defines the orientation of the post when the handrail is on a slope. Select **Always Vertical** to keep the posts vertical even when the handrail is on a slope. Select **Perpendicular to Slope** to keep the posts perpendicular to the slope.

Segment Maximum Spacing

Enter the maximum distance allowed between the posts.

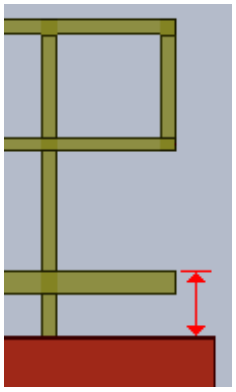


Sloped Segment Maximum Spacing

Enter the maximum distance allowed between posts for that part of a handrail that is on a slope.

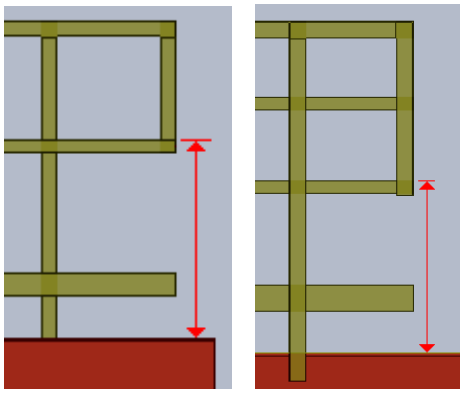
Distance to Top of Toe Plate

Specify the height of the toe plate measured from the path.



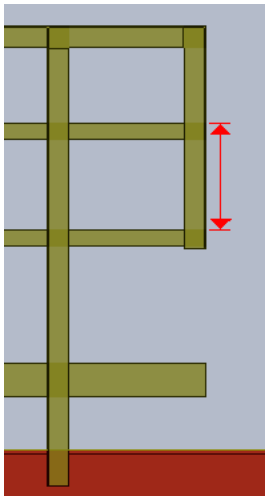
To Top of Midrail Distance

Specify the distance between the path and the top of the mid rail. If you are placing more than one mid rail, this distance is to the top of the bottommost mid rail.



Midrail Spacing

Specify the distance between the mid rails if you are placing more than one mid rail. This value is ignored if you have only one mid rail.



With Post At Turn

Specifies if you want a post in the handrail corner. Select **True** to place a post in the corner. Select **False** to not require a post in the corner.

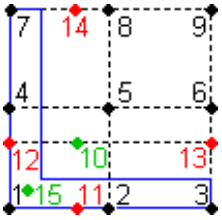
Is Assembly

Not used by the software. Set to **False**.

Is System

Not used by the software. Set to **False**.

Top Rail Section Cardinal Point

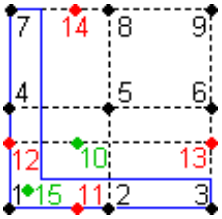


Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Top Rail Section Angle

Defines the angle, in degrees or radians, by which the top rail cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis depending on the member orientation.

Midrail Section Cardinal Point

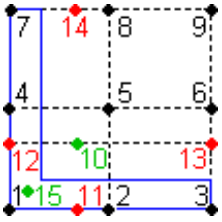


Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Midrail Section Angle

Defines the angle, in degrees or radians, by which the mid rail cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis, depending on the member orientation.

Toe Plate Section Cardinal Point



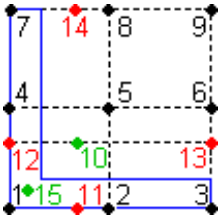
Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Toe Plate Section Angle

Defines the angle, in degrees or radians, by which the toe plate cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis depending on the member orientation.

Post Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.



Post Section Angle

Defines the angle, in degrees or radians, by which the post cross-section is rotated about the member axis. The zero degree position is either the global Z-axis or the global X-axis, depending on the member orientation.

Horizontal Offset

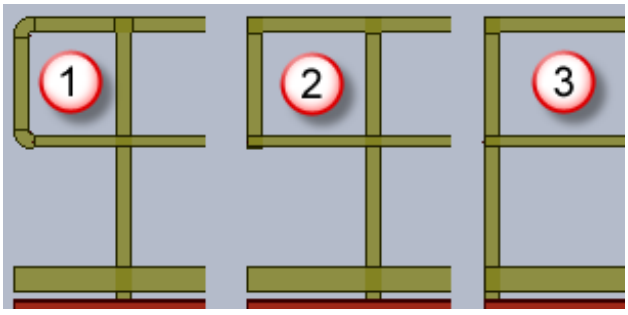
Select the direction the **Horizontal Offset Distance** property value is applied.

Horizontal Offset Distance

Enter the horizontal offset from the defined handrail path to the handrail. You control the direction of the offset using the **Horizontal Offset** property.

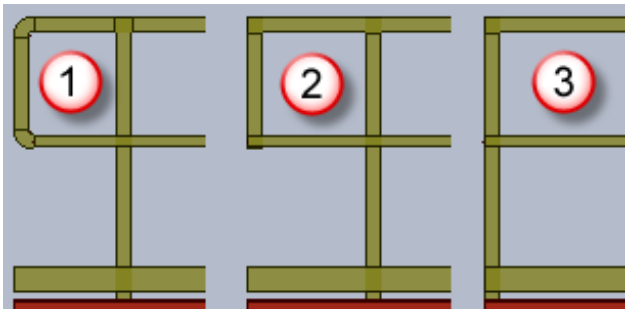
Begin Treatment

Specify a circular (1), rectangular (2), or none (3) end treatment for the beginning of the handrail.



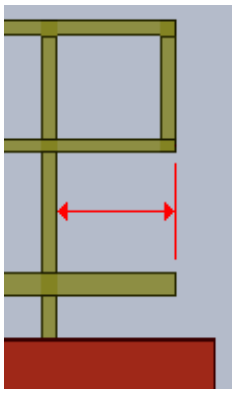
End Treatment

Specify a circular (1), rectangular (2), or none (3) end treatment for the end of the handrail.



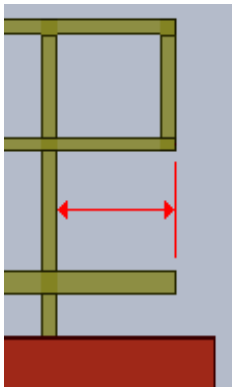
Begin Extension Length

Enter the distance between the beginning of the handrail and the first handrail post. This distance must be greater than zero.



End Extension Length

Enter the distance between the end of the handrail and the last handrail post. This distance must be greater than zero.



Minimum offset Clearance at Post Turn

Enter the minimum distance from a handrail turn that the software can place a post.

Maximum offset Clearance at Post Turn

Enter the maximum distance from a handrail turn that the software can place a post.

Length

Displays the total length of the handrail.

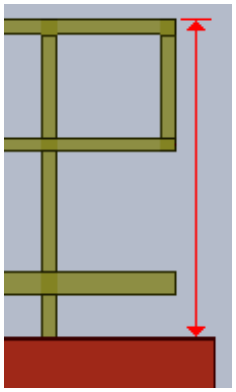
Type A Handrail (TopEmbedded)

Vertical offset value for Embedding

Enter the distance to embed the handrail posts.

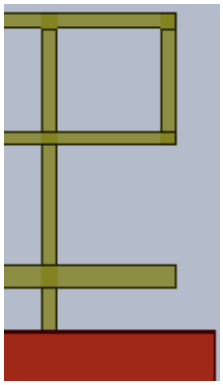
Height

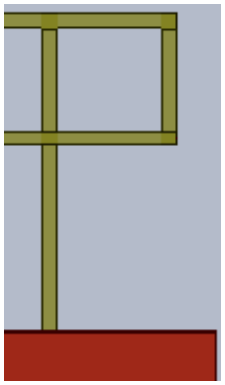
Enter the height of the handrail from the path to the top of the top rail.



With Toe Plate

Select **True** to attach a toe plate to the handrail. Select **False** to place a handrail without a toe plate.





Number of Midrails

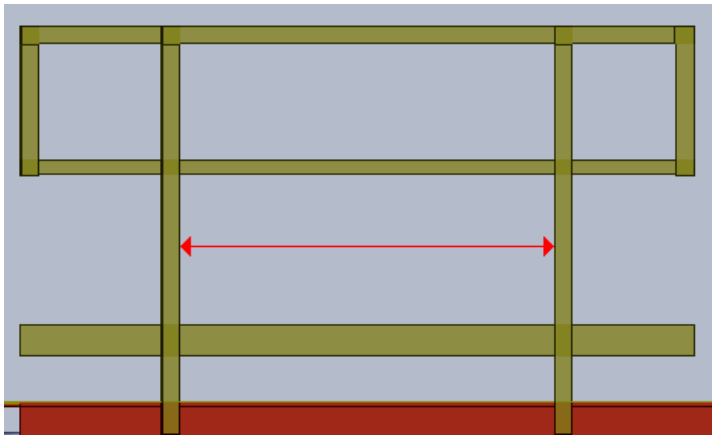
Enter the number of mid rails for the handrail. The first mid rail is placed using the defined **To Top of Mid Rail Distance**. The subsequent mid rails are placed above that using the **Mid Rail Spacing** distance.

Handrail Orientation

Defines the orientation of the post when the handrail is on a slope. Select **Always Vertical** to keep the posts vertical even when the handrail is on a slope. Select **Perpendicular to Slope** to keep the posts perpendicular to the slope.

Segment Maximum Spacing

Enter the maximum distance allowed between the posts.

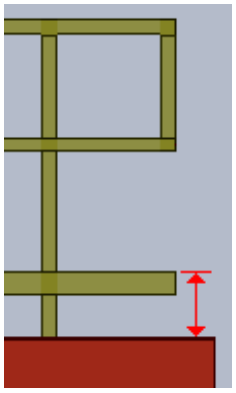


Sloped Segment Maximum Spacing

Enter the maximum distance allowed between posts for that part of a handrail that is on a slope.

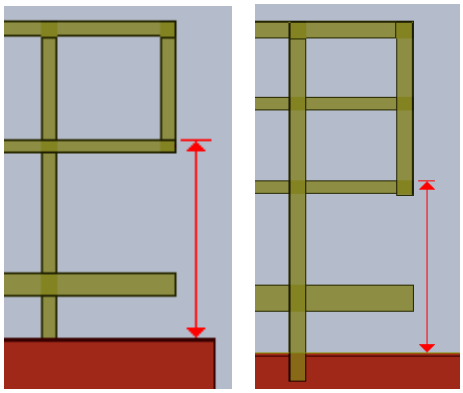
Distance to Top of Toe Plate

Specify the height of the toe plate measured from the path.



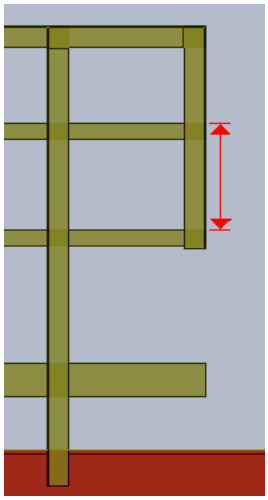
To Top of Midrail Distance

Specify the distance between the path and the top of the mid rail. If you are placing more than one mid rail, this distance is to the top of the bottommost mid rail.



Midrail Spacing

Specify the distance between the mid rails if you are placing more than one mid rail. This value is ignored if you have only one mid rail.



With Post At Turn

Specifies if you want a post in the handrail corner. Select **True** to place a post in the corner. Select **False** to not require a post in the corner.

Is Assembly

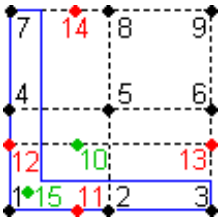
Not used by the software. Set to **False**.

Is System

Not used by the software. Set to **False**.

Top Rail Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

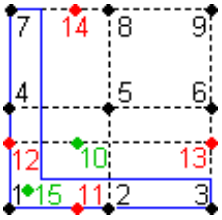


Top Rail Section Angle

Defines the angle, in degrees or radians, by which the top rail cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis depending on the member orientation.

Midrail Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

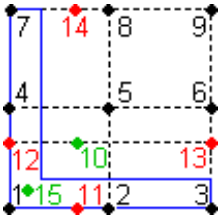


Midrail Section Angle

Defines the angle, in degrees or radians, by which the mid rail cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis, depending on the member orientation.

Toe Plate Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

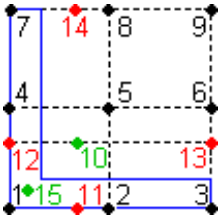


Toe Plate Section Angle

Defines the angle, in degrees or radians, by which the toe plate cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis depending on the member orientation.

Post Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.



Post Section Angle

Defines the angle, in degrees or radians, by which the post cross-section is rotated about the member axis. The zero degree position is either the global Z-axis or the global X-axis, depending on the member orientation.

Horizontal Offset

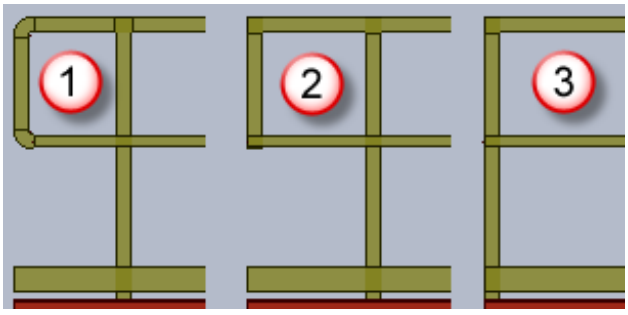
Select the direction the **Horizontal Offset Distance** property value is applied.

Horizontal Offset Distance

Enter the horizontal offset from the defined handrail path to the handrail. You control the direction of the offset using the **Horizontal Offset** property.

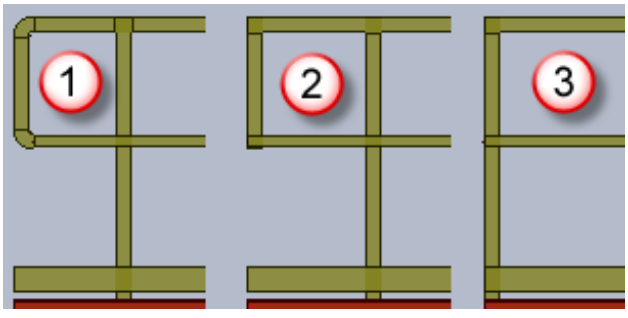
Begin Treatment

Specify a circular (1), rectangular (2), or none (3) end treatment for the beginning of the handrail.



End Treatment

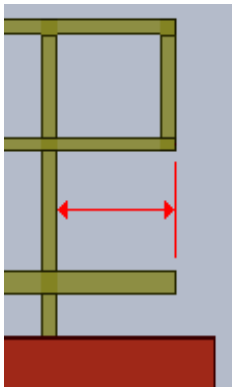
Specify a circular (1), rectangular (2), or none (3) end treatment for the end of the handrail.



Begin Extension Length

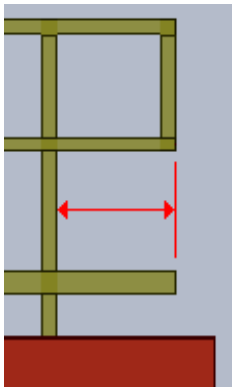
Enter the distance between the beginning of the handrail and the first handrail post. This

distance must be greater than zero.



End Extension Length

Enter the distance between the end of the handrail and the last handrail post. This distance must be greater than zero.



Minimum offset Clearance at Post Turn

Enter the minimum distance from a handrail turn that the software can place a post.

Maximum offset Clearance at Post Turn

Enter the maximum distance from a handrail turn that the software can place a post.

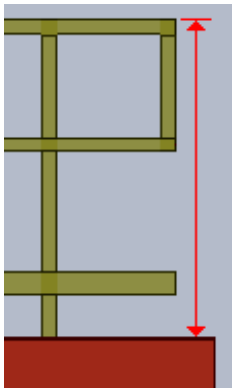
Length

Displays the length of the handrail path.

Type A Handrail (TopMountedonPad)

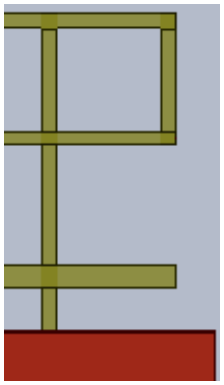
Height

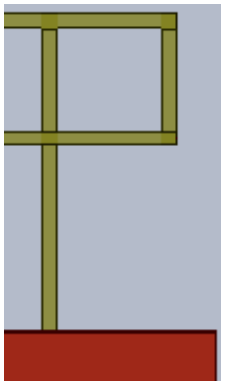
Enter the height of the handrail from the path to the top of the top rail.



With Toe Plate

Select **True** to attach a toe plate to the handrail. Select **False** to place a handrail without a toe plate.





Number of Midrails

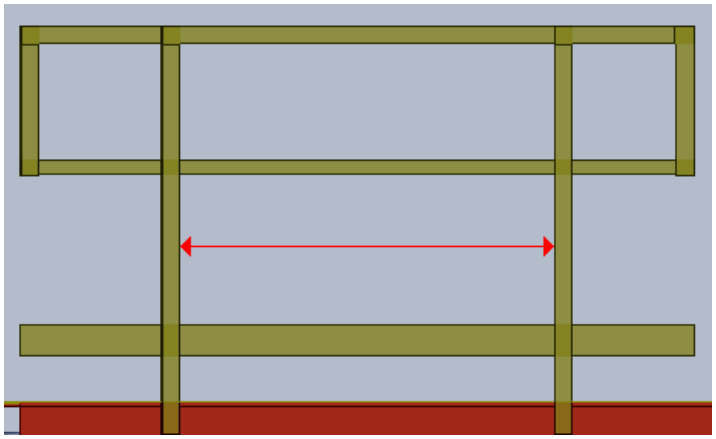
Enter the number of mid rails for the handrail. The first mid rail is placed using the defined **To Top of Mid Rail Distance**. The subsequent mid rails are placed above that using the **Mid Rail Spacing** distance.

Handrail Orientation

Defines the orientation of the post when the handrail is on a slope. Select **Always Vertical** to keep the posts vertical even when the handrail is on a slope. Select **Perpendicular to Slope** to keep the posts perpendicular to the slope.

Segment Maximum Spacing

Enter the maximum distance allowed between the posts.

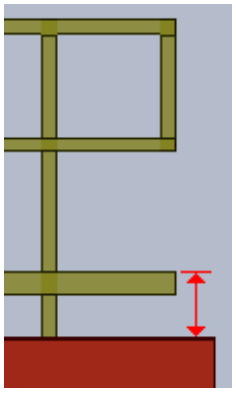


Sloped Segment Maximum Spacing

Enter the maximum distance allowed between posts for that part of a handrail that is on a slope.

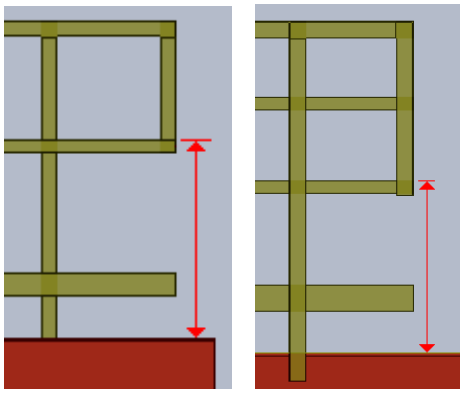
Distance to Top of Toe Plate

Specify the height of the toe plate measured from the path.



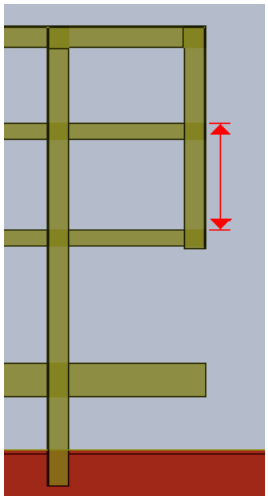
To Top of Midrail Distance

Specify the distance between the path and the top of the mid rail. If you are placing more than one mid rail, this distance is to the top of the bottommost mid rail.



Midrail Spacing

Specify the distance between the mid rails if you are placing more than one mid rail. This value is ignored if you have only one mid rail.



With Post At Turn

Specifies if you want a post in the handrail corner. Select **True** to place a post in the corner. Select **False** to not require a post in the corner.

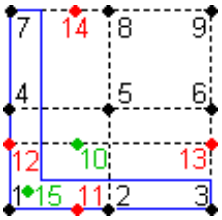
Is Assembly

Not used by the software. Set to **False**.

Is System

Not used by the software. Set to **False**.

Top Rail Section Cardinal Point

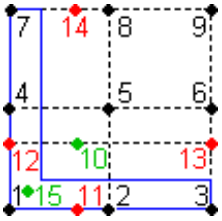


Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Top Rail Section Angle

Defines the angle, in degrees or radians, by which the top rail cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis depending on the member orientation.

Midrail Section Cardinal Point

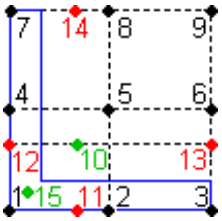


Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Midrail Section Angle

Defines the angle, in degrees or radians, by which the mid rail cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis, depending on the member orientation.

Toe Plate Section Cardinal Point



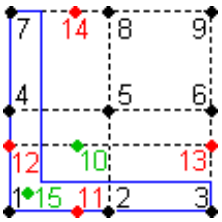
Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Toe Plate Section Angle

Defines the angle, in degrees or radians, by which the toe plate cross-section is rotated about the member axis. The zero-degree position is either the global Z-axis or the global X-axis depending on the member orientation.

Post Section Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.



Post Section Angle

Defines the angle, in degrees or radians, by which the post cross-section is rotated about the member axis. The zero degree position is either the global Z-axis or the global X-axis, depending on the member orientation.

Horizontal Offset

Select the direction the **Horizontal Offset Distance** property value is applied.

Horizontal Offset Distance

Enter the horizontal offset from the defined handrail path to the handrail. You control the direction of the offset using the **Horizontal Offset** property.

Vertical Offset value for Pad

Enter the height of the mounting pad.

Pad Plate Length

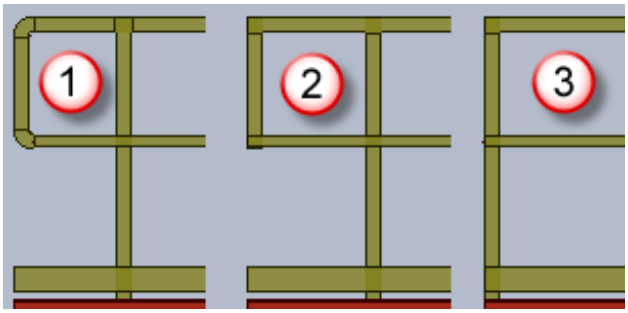
Enter the length of the pad plate.

Pad Plate Width

Enter the width of the pad plate.

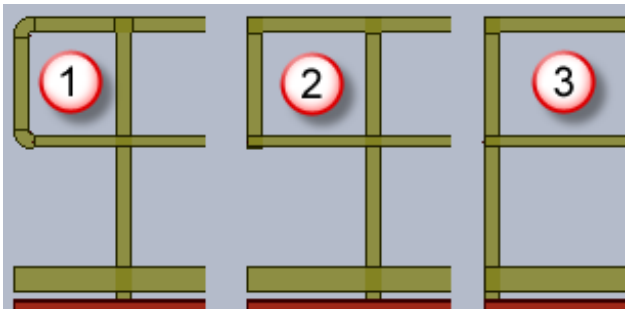
Begin Treatment

Specify a circular (1), rectangular (2), or none (3) end treatment for the beginning of the handrail.



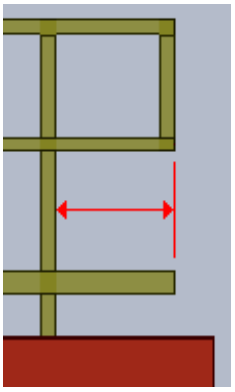
End Treatment

Specify a circular (1), rectangular (2), or none (3) end treatment for the end of the handrail.



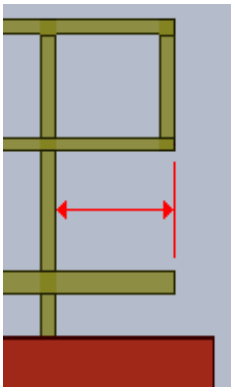
Begin Extension Length

Enter the distance between the beginning of the handrail and the first handrail post. This distance must be greater than zero.



End Extension Length

Enter the distance between the end of the handrail and the last handrail post. This distance must be greater than zero.



Minimum offset Clearance at Post Turn

Enter the minimum distance from a handrail turn that the software can place a post.

Maximum offset Clearance at Post Turn

Enter the maximum distance from a handrail turn that the software can place a post.

Length

Displays the length of the handrail path.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material density multiplied by the object's solid volume. Therefore, the material of the object affects the weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight

Specifies the dry weight of the object.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Dry WCG Origin

Specifies the way in which the dry weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the dry weight center-of-gravity location relative to the active coordinate system.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Surface Treatment and Coating

Exterior Coating Requirement

Specifies the coating requirement for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Exterior Coating Area

Specifies the area of the coating for the object.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

See Also

Handrail Properties Dialog Box (on page 422)

Definition Tab (Handrail Properties Dialog Box)

The **Definition** tab displays the handrail properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one handrail and then selected the properties command, only the common properties between the selected handrails display.

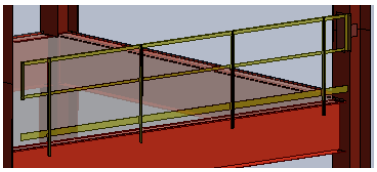
The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties. You can access this document using the **Help > Printable Guides** command in the software.

See Also

Handrail Properties Dialog Box (on page 422)

SECTION 28

Place Handrails by Member



Places a handrail that is associated with a member part's frame connections. Because the handrail is associated with the member part's frame connections, the handrail length automatically resizes if the member part length changes. You can also define clearances between the handrail-end and the member-end or a column located at the member end.

You can use this command to place a handrail on a single member or multiple members at a time. If you select multiple members, each handrail placed is independent of the others after placement.

If you split a member after placing a handrail on it, the handrail will still be associated to the original placement frame connections, one on either side of the split. If the two new member parts (created by the split) need their own handrails, you will need to delete the original handrail and then place the two new handrails, one on each new member part.

Place Handrail by Member Ribbon

Specifies the properties for the handrail that you are placing or editing.

Handrail Properties

Activates the **Handrail Properties** dialog box. You can use this dialog box to specify additional handrail properties, such as rail section and toe plate section, which you cannot set on the ribbon. For more information, see *Handrail Properties Dialog Box* (on page 422).

Select Members

Select the members on which to place a handrail.

Walking Surface

Select the surface on which people will be walking. The software using this point to know on which side of the member to place the handrail and to calculate the toe-plate and railing height.

Convert

Decomposes the handrail symbol to individual structural members for each post and rail. You should convert a handrail when you want to add, edit, or delete the positions of individual posts, rails, or perform some other handrail customization that cannot be done using the handrail symbol properties. The Handrail object will become the system parent of the individual members. Deleting the Handrail object will delete the children members. Handrails that you have converted to member parts cannot be converted back to a handrail symbol. You can place handrails as individual members during initial placement by clicking **Convert** instead of **Finish**, or you can select an existing handrail and convert it to member parts.

Finish

Places the handrail in the model.

Cancel

Clears your selections from the selection set.

Accept

Confirms your selections as the ones to process.

System

Select the system to which the handrail belongs. You can create new systems in the Systems and Specifications task.

Type

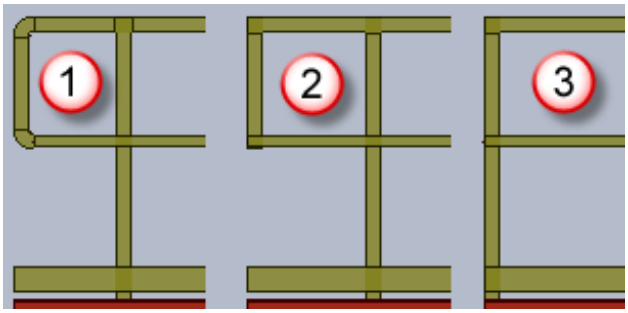
Select the type of handrail to place from the catalog.

Name

Specify the name for the handrail that you are placing. This box is read-only unless you are using the **User Defined** naming rule.

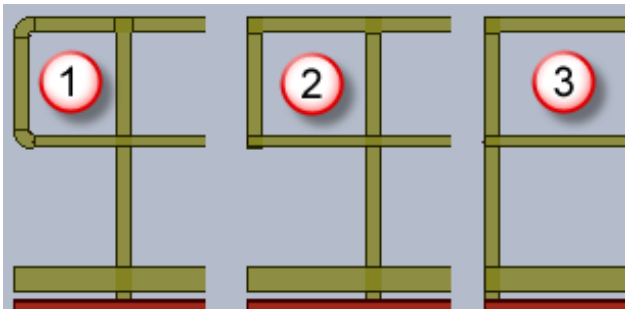
Beginning Treatment

Specify a circular (1), rectangular (2), or none (3) end treatment for the beginning of the handrail.



End Treatment

Specify a circular (1), rectangular (2), or none (3) end treatment for the end of the handrail.

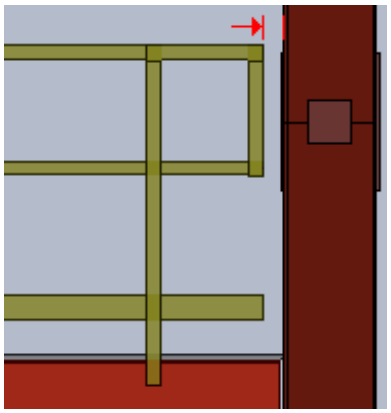


Offset Reference

Select how the horizontal offset of the handrail is measured from the member. You can edit the horizontal offset value by editing the **Horizontal Path Offset Distance** value in the handrail Standard category properties.

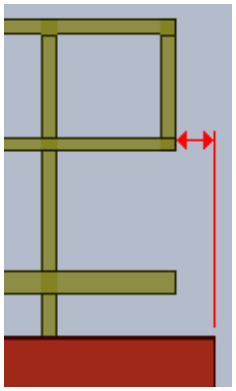
Column Clearance

Enter the distance between the column, if one exists, and the handrail. If a column does not exist, this value is ignored and the **End Clearance** value is used instead.



End Clearance

Enter the distance between the member end and the handrail end. This value is only used when a column does not exist at that member end.



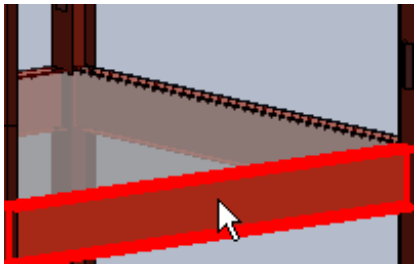
What do you want to do?

- *Place a handrail by member* (on page 461)
- *Split a handrail* (on page 421)
- *Edit handrail placement* (on page 421)
- *Edit handrail properties* (on page 421)
- *Convert handrail to individual members* (on page 421)
- *Delete a handrail* (on page 422)

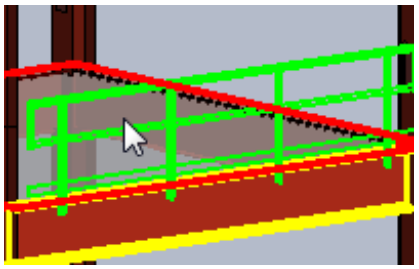
Place a handrail by member

1. Click **Place Handrail by Member** on the vertical toolbar.
2. Select the SideMountedtoMember handrail using the **Type** option.

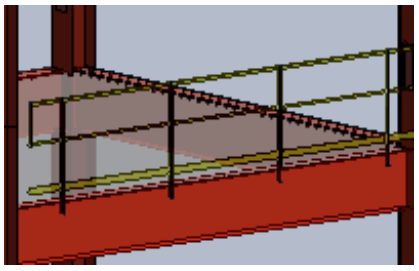
   TIP You can place other handrail types, but only SideMountedtoMember provides the column and end clearance options.
3. Select the members for the handrail.



4. Click **Accept**
5. In the graphic view, click on the walking surface side of the member.



6. Enter a distance for the column clearance or the end clearance as needed.
7. Click **Finish**.



Edit handrail placement

1. Click **Select** on the vertical toolbar.
2. Select **Handrails** in the **Locate Filter**.
3. Select the handrail to edit.
4. Using the ribbon option, edit the handrail placement as needed.

Edit handrail properties

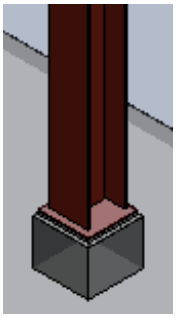
1. Click **Select** on the vertical toolbar.
2. Select **Handrails** in the **Locate Filter**.
3. Select the handrail to edit.
4. Click **Edit > Properties**.
5. Edit the properties as needed. For more information about these properties, see *Handrail Properties Dialog Box* (on page 422).

Delete a handrail

1. Click **Select** on the vertical toolbar.
2. Select **Handrails** in the **Locate Filter**.
3. Select the handrail to delete.
4. Press **Delete**.

SECTION 29

Place Footings



Places column footings in the model. You can place footings before or after columns are placed in the model. The software creates a relationship between the footing and the member end to which the footing connects.

There are two types of footings that you can place: single and combined. Single footings create a one-to-one relationship between the column and the footing. Combined footings create a many-to-one relationship between a multiple columns and a single footing.

The bottom elevation of the footing can optionally be associated with a reference plane. If the reference plane moves, the footing height adjusts automatically to the new elevation of the plane. Not all footing types support associated planes. Footing types that do not support associated planes have a **Number of Supporting** value of zero in the catalog.

A footing is used to transfer the loads of a column to the ground or sub-structure and to provide mounting stability for the supported column. A typical footing is comprised of:

- A base that supports one or more piers. Some footings for light-weight structures, such as a light pole, do not have a base.
- A pier that rests on the base and supports the grout layer. Some footings do not have a pier in which case the grout is put directly on the base.
- A grout layer that rests on the pier and supports the base plate.
- A base plate that rests on the grout layer and is welded to the supported column.

You can place base plates using the *Place Assembly Connection* (on page 174).

Helpful things to know about footings:

- Combined footings require a minimum of two columns for placement. If you remove columns from a combined footing such that a single column remains, the software sends the combined footing to the To Do List.
- When placing a combined footing with a merged pier, all columns must have the same bottom elevation. The software will not allow a column with a different bottom elevation to be added to the merged pier.
- The software places key points and control points that can be selected as layout points in drawings on combined footings. Key points are placed at the vertices of slabs and piers. Control points are placed at column locations.

Place Footing Ribbon

Specifies the properties for the footing that you are placing or editing.

Properties

Activates the **Footing Properties** dialog box. You can use this dialog box to specify additional properties, such as material and material grade, which you cannot set on the

ribbon. For more information, see *Footing Properties Dialog Box* (on page 473).

Select Member

If **Placement** is set to **By Member**, select the column, or columns, in the model for which to place footings. When you are done selecting columns, click **Accept** ✓. If you accidentally select a column that you did not want, click **Cancel** ✗, and then re-select the columns.

If **Placement** is set to **By Point**, identify the point, or points, in the model at which you want a footing. The point identifies the bottom elevation location of a future column.

Select Bottom Plane

Select the bottom plane, or supporting surface, for the footing. The software disables this option if the footing type that you have selected does not require that a supporting surface be defined.

Finish

Places the footing in the model.

Cancel

Rejects the objects that you have selected.

Accept

Confirms the objects that you have selected.

Placement

Select **By Member** to place footings at the bottom of members that you specify. Select **By Point** to place footings at a point you specify in the model. Use the **By Point** option when you want to place footings before the columns in the model.

System

Select the system in which to place the footing. You can create new systems in the Systems and Specifications task.

Type

Select the type of footing to place. Select **More** to select from all available footings in the catalog. If you start to select columns and then change the footing type, the software clears the selected columns because most footing types require different column orientations and the columns that you selected may not work with the new footing type.

Name

Type a name for the footing.

Plane Method

Specifies how the bottom elevation of the footing is defined. This option is available only when you select a footing type from the catalog that has a **Number of Supporting** value other than zero.

- Select **Associative** to associate the bottom of the footing to a plane that you select. If that plane moves, the bottom of the footing adjusts accordingly.
- Select **Elevation Value** to enter the bottom elevation in the **Elevation Value** box.

Elevation Value

Defines the bottom elevation of the footing relative to the global origin. You can also select a plane in the model to define this elevation; however, no association is made to this plane. This option is available only when **Plane Method** is set to **Elevation Value**.

Select Footing Dialog Box

Allows selection of the type of footing to be placed. This dialog box appears when you select **More** in the **Type** list. By browsing through the hierarchy, you can find any footing in the Catalog database. After you select a footing, the software returns you to the model, where you can finalize placement.

 **Back**

Returns you to the previously selected footing type or node. Use this command to navigate through the hierarchy to the specific type that you need.

 **Forward**

Sends you to the last selected footing type or node that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific type that you need.

 **Up One Level**

Brings up the next highest level of the catalog hierarchy. Use this command to navigate through the hierarchy to the specific type that you need.

 **Properties**

Displays the properties of the selected footing. Because you cannot modify any properties until the footing is placed, all properties on the dialog box are read-only.

 **Preview**

Displays a picture of the selected footing. The image file must be assigned to the footing in the reference data.

 **List View**

Sets the dialog box to display footings in a list view.

 **Grid View**

Sets the dialog box to display footings in a spreadsheet-style grid view.

Address

Specifies your exact location within the displayed hierarchy.

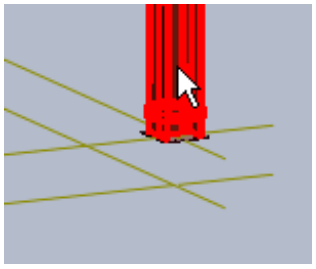
What do you want to do?

- *Place a single footing* (on page 466)
- *Place a single footing by point* (on page 467)
- *Place a single footing without an associated plane* (on page 468)
- *Place a combined footing* (on page 469)
- *Place a combined footing by point* (on page 470)

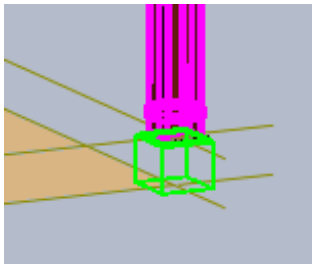
- *Add column to a combined footing* (on page 470)
- *Remove column from a combined footing* (on page 471)
- *Edit footing properties* (on page 472)
- *Delete a footing* (on page 472)

Place a single footing

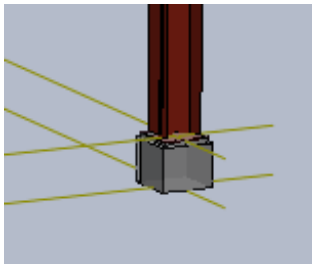
1. Click **Place Footing** on the vertical toolbar.
2. Set **Placement** to **By Member**.
3. Select the system for the footing.
4. Using the **Type** option, select the footing type to place.
5. Select the column where the footing is to be placed.



6. Click **Accept** ✓.
7. Select the supporting plane if the footing type that you are placing requires it, and then click **Accept** ✓.

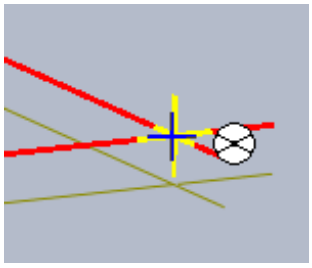


8. Click **Finish**.

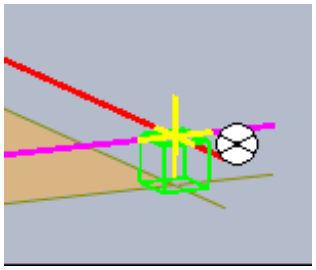


Place a single footing by point

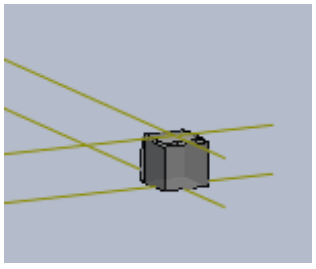
1. Click **Place Footing** on the vertical toolbar.
2. Set **Placement** to **By Point**.
3. Select the system for the footing.
4. Using the **Type** option, select the footing type to place.
5. Identify a point in the model where you want the footing. The point identifies the bottom elevation of a future column.



6. Click **Accept** ✓.
7. Select the supporting plane if the footing type that you are placing requires it, and then click **Accept** ✓.

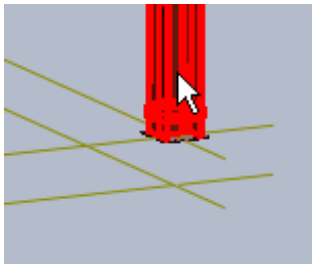


8. Click **Finish**.



Place a single footing without an associated plane

1. Click **Place Footing** on the vertical toolbar.
2. Set **Placement** to **By Member**.
3. Select the system for the footing.
4. Using the **Type** option, select the footing type to place. You must select a footing that has a **Number of Supporting** value other than zero.
5. Select the column for the footing.



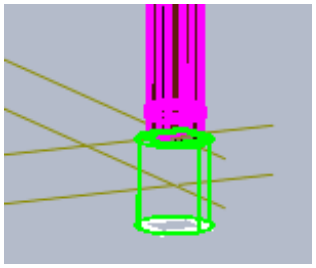
6. Click **Accept** ✓.
7. Set **Plane Method** to **Elevation Value**.



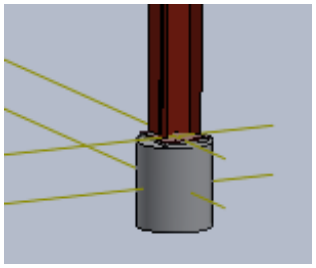
8. Select a plane in the model to define the bottom elevation of the footing.

   -OR-

   Type an elevation relative to the global origin in the **Elevation Value** box.
9. Click **Accept** ✓.

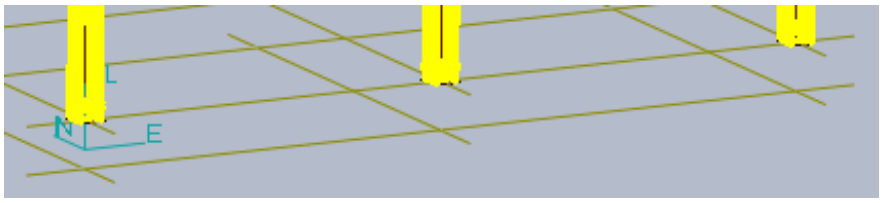


10. Click **Finish**.

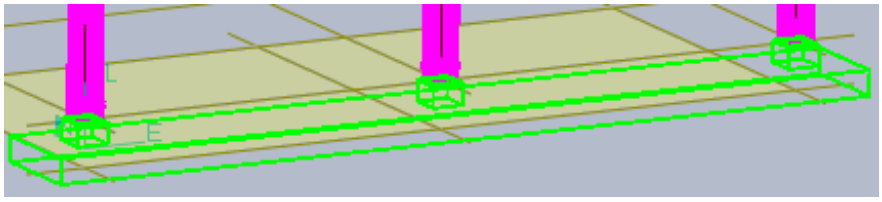


Place a combined footing

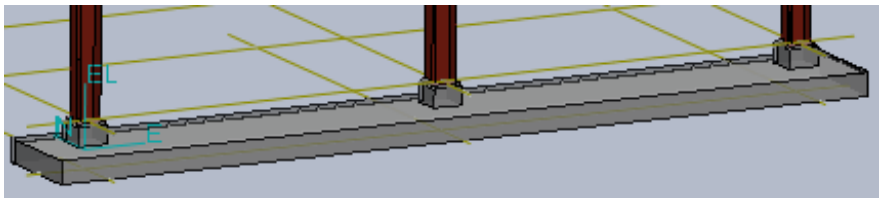
1. Click **Place Footing** on the vertical toolbar.
2. Set **Placement** to **By Member**.
3. Select the system for the footing.
4. Using the **Type** option, select the combined footing type to place.
5. Select the columns where the footing is to be placed. You must select at least two columns.



6. Click **Accept** ✓ when you are done selecting the columns.
7. Select the supporting plane if the footing type that you are placing requires it, and then click **Accept** ✓.

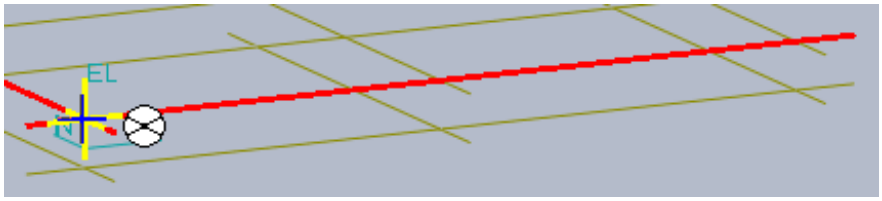


8. Click **Finish**.

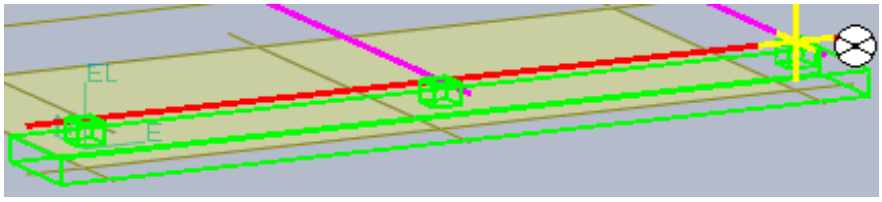


Place a combined footing by point

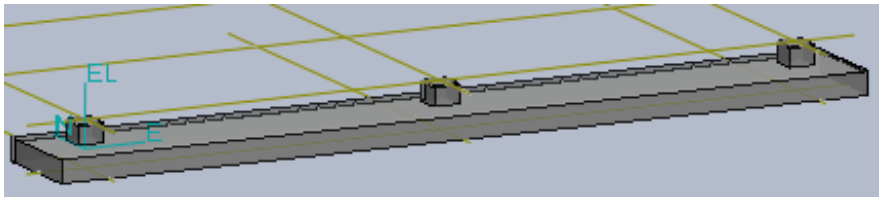
1. Click **Place Footing** on the vertical toolbar.
2. Set **Placement** to **By Point**.
3. Select the system for the footing.
4. Using the **Type** option, select the footing type to place.
5. Identify points in the model where you want the footing. You must identify at least two points.



6. Click **Accept** when you are done.
7. Select the supporting plane if the footing type that you are placing requires it, and then click **Accept**.

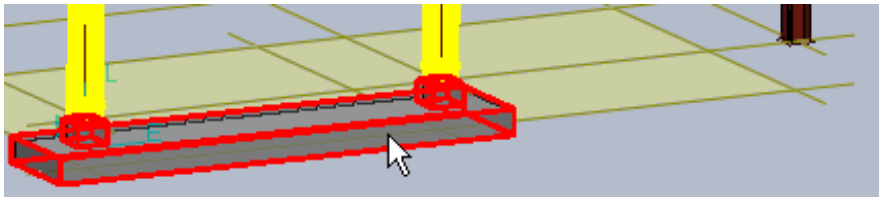


8. Click **Finish**.

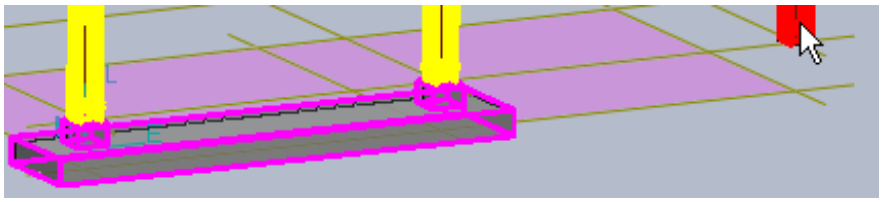


Add column to a combined footing

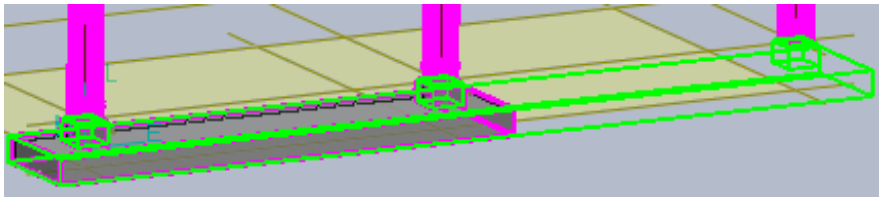
1. Click **Select** on the vertical toolbar.
2. Select **Footings** in the **Locate Filter**.
3. Select the footing to which you want to add the column.



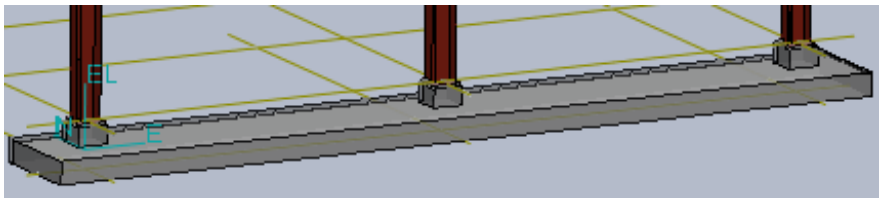
4. Click **Select Member** on the ribbon bar.
5. Select the column, or columns, to add to the footing.



6. Click **Accept** when you are done selecting columns.

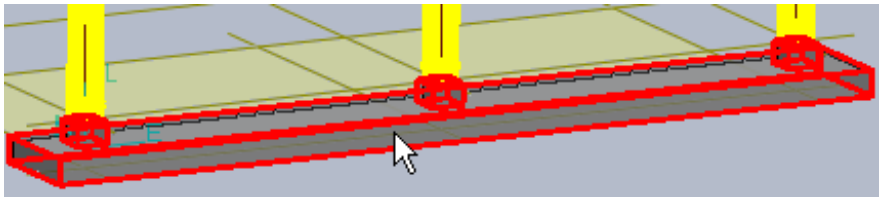


7. Click **Finish**.

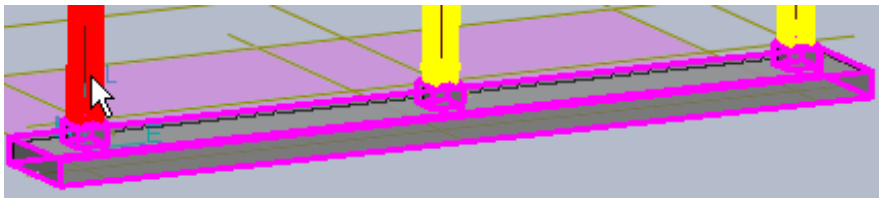


Remove column from a combined footing

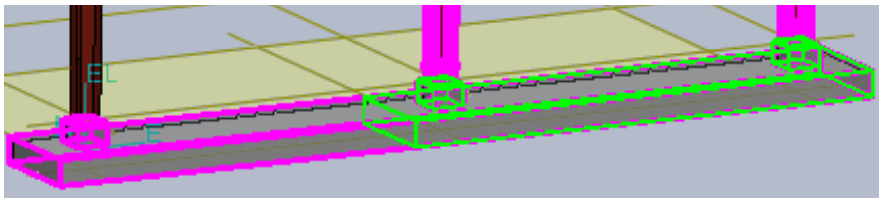
1. Click **Select** on the vertical toolbar.
2. Select **Footings** in the **Locate Filter**.
3. Select the footing from which you want to remove the column.



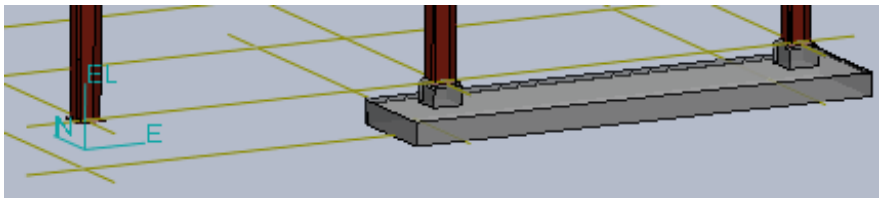
4. Click **Select Member** on the ribbon bar.
5. Select the column, or columns, to remove from the footing.



6. Click **Accept** ✓ when you are done selecting columns.



7. Click **Finish**.



NOTE Combined footings must support a minimum of two columns. Combined footings with only a single column are sent to the To Do List.

Edit footing properties

1. Click **Select** on the vertical toolbar.
2. Select **Footings** in the **Locate Filter**.
3. Select the footing to edit.
4. Click **Edit > Properties**.
5. Edit the properties as needed.

Delete a footing

1. Click **Select** on the vertical toolbar.
2. Select **Footings** in the **Locate Filter**.
3. Select the footing to delete.
4. Click **Delete**.

NOTE The software automatically deletes a single footing when you delete the associated column.

Footing Properties Dialog Box

Specifies the properties for the footing that you are editing.

See Also

Definition Tab (Footing Properties Dialog Box) (on page 478)
Occurrence Tab (Footing Properties Dialog Box) (on page 473)
Place Footings (on page 463)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

Occurrence Tab (Footing Properties Dialog Box)

The **Occurrence** tab displays the footing properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one footing, and then selected the properties command, only the common properties between the selected footings display.

When viewing properties for a single footing, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on properties. You can access this document using the **Help > Printable Guides** command in the software.

Category

Select the properties that you want to view for the footing. Footing properties are divided into several different categories: Standard and Weight and CG. You select the category to define values for by using the **Category** option.

Standard

The standard properties that display depend on the reference data for the footing. Because footing reference data is fully customizable, only the common properties are documented.

Name

Displays the name of the footing. The footing name is based on the **Name Rule** selection. If you want to type a new name for the footing, in the **Name Rule** box, select **User Defined**, and then type a name for the footing in the **Name** box.

Name Rule

Specify the naming rule that you want to use to name this footing.

- **Default Name Rule** - Names the footing using the format "<footing type>-<location>-<index>" where <footing type> is what you select in the **Type** box on the ribbon, <location> is the global workshare location ID, and <index> is a unique index number that starts at 0001. For example, RectSlabFootingAsm-1-0043.
- **Unique Name Rule** - Names the footing using the format "<system>-<footing type>-<location>-<index>" where <system> is the name of the parent system to which the footing belongs, <footing type> is what you select in the **Type** box on the ribbon, <location> is the global workshare location ID, and <index> is a unique index number that starts at 0001. For example, Structure System-RectSlabFootingAsm-1-0043.
- **User Defined** - Select to specify the footing name yourself in the **Name** box.

System

Displays the name of the parent system. You can create new systems in the **Systems and Specifications** task.

Grout Component

The symbol used to represent the grout component.

Pier Component

The symbol used to represent the pier component.

Slab Component

The symbol used to represent the slab component.

Pier Material

Select the pier material. Available materials are defined in the **Catalog** task.

Pier Grade

Select the pier material grade. Available material grades are defined in the **Catalog** task.

Slab Material

Select the slab material.

Slab Grade

Select the slab material grade.

Grout Material

Select the grout material.

Grout Grade

Select the grout material grade.

Detail Reference

Enter the detail drawing number to reference.

With Grout Pad

Specifies whether or not a grout pad is included between the pier and the member being supported.

Reporting Requirement

Specifies whether or not this footing is reported.

Reporting Type

Select whether the footing is to be tracked by the material control system or not.

Pier Length

Displays or specifies the pier length. If this box is read-only and you want to specify the length yourself, then you need to set **Pier Sizing Rule** to **User Defined**.

Pier Width

Displays or specifies the pier width. If this box is read-only and you want to specify the width yourself, then you need to set **Pier Sizing Rule** to **User Defined**.

Pier Height

Specifies the distance between the top of the supporting slab and the bottom of the grout (if there is a grout pad) or the bottom of the supported member (if there is no grout pad.)

Pier Shape

Displays the pier shape.

Pier Sizing Rule

Select how you want the size of the pier to be calculated.

- Select **Clearance** to have the software size the pier length and width based on either the supported member's cross-section size or the grout size if grout is present plus a clearance value that you specify. The maximum range of the cross-section or grout is automatically calculated and the pier length and pier width values are set using the cross-section range plus the value you specify in **Pier Edge Clearance**.
- Select **User Defined** to specify the pier length and width values yourself.

Pier Orientation

Select whether the pier should be oriented relative to the local coordinate system or the global coordinate system.

Pier Rotation Angle

Specify the rotation angle relative to the north (or y-axis) of the coordinate system selected in **Pier Orientation**.

Pier Edge Clearance

Specify the distance between the side of the pier and the maximum range of either the supported member's cross-section size or the grout size for that side. This value is used when **Pier Sizing Rule** is set to **Clearance**.

Pier Size Increment

Specify the step size to increase the pier length and width dimensions when the software calculates them. For example, if the supported member's cross-section range is 18.25 cm by 23.5 cm and you specify a 1 cm size increment, then the software would round up the pier dimensions to 19 cm by 24 cm. If you were to specify a 0.5 cm size increment, then the pier dimensions would be 18.5 cm by 23.5 cm.

Pier Chamfered

Specifies if the corners of square piers are chamfered or not.

Pier Chamfer Size

Specifies the size of the chamfer if Pier Chamfered is set to yes.

Grout Length

Displays the grout length. If this box is read-only and you want to specify the length yourself, then you need to set **Grout Sizing Rule** to **User Defined**.

Grout Width

Displays the grout width. If this box is read-only and you want to specify the width yourself, then you need to set **Grout Sizing Rule** to **User Defined**.

Grout Height

Specifies the grout height, which is the distance between the top of the pier and the bottom of the supported member.

Volume

Displays the calculated volume of the footing (including all of the components).

Slab Length

Displays or specifies the slab length. If this box is read-only and you want to specify the length yourself, then you need to set **Slab Sizing Rule** to **User Defined**.

Slab Width

Displays or specifies the slab width. If this box is read-only and you want to specify the width yourself, then you need to set **Slab Sizing Rule** to **User Defined**.

Slab Height

Specifies the distance between the bottom of the slab and the bottom of the pier.

Slab Shape

Displays the slab shape.

Slab Sizing Rule

Select how you want the size of the slab to be calculated.

- Select **Clearance** to have the software size the slab length and width based on the pier size plus a clearance value that you specify. The maximum range of the pier is automatically calculated and the slab length and slab width values are set using that range plus the value you specify in **Slab Edge Clearance**.
- Select **User Defined** to specify the slab length and width values yourself.

Slab Orientation

Select whether the slab should be oriented relative to the local coordinate system or the global coordinate system.

Slab Rotation Angle

Specify the rotation angle relative to the north (or y-axis) of the coordinate system selected in **Slab Orientation**.

Slab Edge Clearance

Specify the distance between the side of the slab and the maximum range of the pier for that side. This value is used when **Slab Sizing Rule** is set to **Clearance**.

Slab Size Increment

Specify the step size to increase the slab length and width dimensions when the software calculates them. For example, if the slab is 18.25 cm by 23.5 cm and you specify a 1 cm size increment, then the software would round up the slab dimensions to 19 cm by 24 cm. If you were to specify a 0.5 cm size increment, then the slab dimensions would be 18.5 cm by 23.5 cm.

Surface Area

Displays the total outside surface area of the slab, pier, and grout.

Grout Shape

Select the grout shape that you want to use.

Grout Sizing Rule

Select how you want the size of the grout to be calculated.

- Select **Clearance** to have the software size the grout length and width based on the supported member's cross-section size plus a clearance value that you specify. The maximum range of the cross-section is automatically calculated and the grout length and grout width values are set using the cross-section range plus the value you specify in **Grout Edge Clearance**.
- Select **User Defined** to specify the grout length and width values yourself.

Grout Orientation

Select whether the grout should be oriented relative to the local coordinate system or the global coordinate system.

Grout Rotation Angle

Specify the rotation angle relative to the north (or y-axis) of the coordinate system selected in **Grout Orientation**.

Grout Edge Clearance

Specify the distance between the side of the grout and the maximum range of the supported member's cross-section for that side. This value is used when **Grout Sizing Rule** is set to **Clearance**.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material density multiplied by the object's solid volume. Therefore, the material of the object affects the weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight

Specifies the dry weight of the object.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Dry WCG Origin

Specifies the way in which the dry weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the dry weight center-of-gravity location

relative to the active coordinate system.

See Also

Footing Properties Dialog Box (on page 473)

Definition Tab (Footing Properties Dialog Box)

The **Definition** tab displays the footing properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one footing and then selected the properties command, only the common properties between the selected footings display.

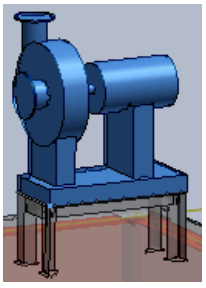
The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties. You can access this document using the **Help > Printable Guides** command in the software.

See Also

Footing Properties Dialog Box (on page 473)

SECTION 30

Place Equipment Foundations



Places equipment foundations in the model. An equipment foundation makes the installation of equipment easier. The foundation is typically connected to the floor, deck, or wall, and the equipment is then bolted to the foundation.

When you place foundations from the catalog, many properties of the foundations are already defined in the reference data. However, the software calculates some of the properties, including the length of its legs, when you place the foundation in the model. For example, when you select a piece of equipment, the software matches the bolt hole patterns of the foundation and the foundation port and automatically orients and sizes the foundation and its legs accordingly.

You can place the foundation that is used to support the equipment either before or after you have placed the equipment in the model.

When you place equipment in the model, you position it in space as required to serve the design needs. Each piece of equipment is expected to have at least one foundation port with two or more holes. The foundation port determines the placement of the foundation to appropriately support the equipment. The foundation is then designed, using the bolt hole pattern from the foundation port of the equipment, to support the equipment in the functionally required position.

Place Equipment Foundation Ribbon

Specifies the properties for the equipment foundation that you are placing or editing.

Properties

Activates the **Equipment Foundation Properties** dialog box. You can use this dialog box to specify additional properties, such as material and material grade, which you cannot set on the ribbon. For more information, see *Equipment Foundation Properties Dialog Box* (on page 483).

Placement

Select **By Equipment** to place the equipment foundation by selecting an equipment or equipment foundation port. Select **By Point** to place the equipment foundation at a selected point in space.

Select Equipment

Select the equipment for which to place a foundation. You can select more than one piece of equipment to place a single foundation under all the selected equipment.

Select Support Surface

Select the surface to support the foundation, and thus the equipment.

Finish

Places the foundation.

Cancel

Rejects the selected object.

Accept

Confirms that the selected object is the object that you want to use.

By Rule

Select to allow the software to select the equipment foundation to use based on the default foundation defined for the equipment part. The default foundation for the equipment appears in the **Type** box. The default foundation for the equipment part is defined in the equipment reference data. The foundation name specified in the equipment reference data must exactly match an equipment foundation part defined in the equipment foundation reference data.

System

Select the system in which to place the foundation.

Type

Select the type of foundation to place. Select **More..**. to select from all available equipment foundations in the catalog.

Name

Type a name for the foundation.

Select Equipment Foundation Dialog Box

Allows selection of the type of equipment foundations to be placed. This dialog box appears when you select **More** in the **Type** list. By browsing through the hierarchy, you can find any equipment foundation in the Catalog database. After you select an equipment foundation, the software returns you to the model, where you can finalize placement.

Back

Returns you to the previously selected equipment foundation type or node. Use this command to navigate through the hierarchy to the specific type that you need.

Forward

Sends you to the last selected equipment foundation type or node that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific type that you need.

Up One Level

Brings up the next highest level of the catalog hierarchy. Use this command to navigate through the hierarchy to the specific type that you need.

Properties

Displays the properties of the selected equipment foundation. Because you cannot modify any properties until the equipment foundation is placed, all properties on the dialog box are read-only.

Preview

Displays a picture of the selected equipment foundation. The image file must be assigned to

the equipment foundation in the reference data.

List View

Sets the dialog box to display equipment foundations in a list view.

Grid View

Sets the dialog box to display equipment foundations in a spreadsheet-style grid view.

Address

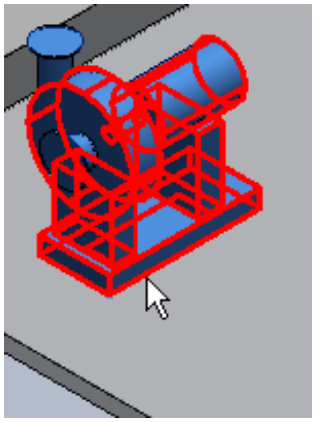
Specifies your exact location within the displayed hierarchy.

What do you want to do?

- *Place an equipment foundation (By Equipment)* (on page 481)
- *Place an equipment foundation (By Point)* (on page 482)
- *Edit an equipment foundation* (on page 483)
- *Edit equipment foundation properties* (on page 483)
- *Delete an equipment foundation* (on page 483)

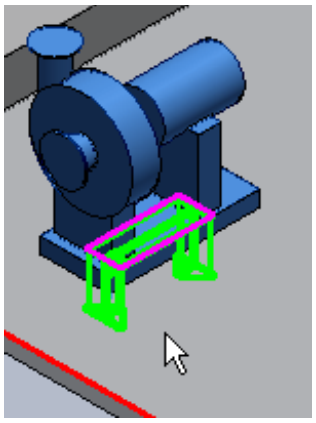
Place an equipment foundation (By Equipment)

1. Click **Place Equipment Foundation** on the vertical toolbar.
2. Select **By Equipment** as the placement type.
3. Select the equipment for which to place a foundation.

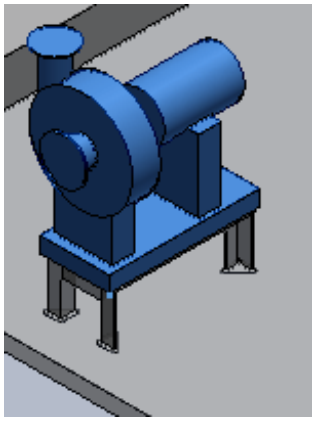


4. Click **Accept** ✓.

5. Select the supporting plane for the equipment foundation.



6. Click **Accept** ✓.
7. Verify that the **Type** and **System** options are set to your satisfaction. Clear the **By Rule** option if you want to manually select the equipment foundation that you want.
8. Click **Finish**.



Place an equipment foundation (By Point)

1. Click **Place Equipment Foundation** on the vertical toolbar.
2. Select **By Point** as the placement type.
3. Select the type of foundation you want to place.
4. Select the system in which to place the foundation.
5. Click on a point in the model where you want to place the foundation.
6. Select the supporting plane for the equipment foundation.
7. Click **Accept** ✓.
8. Verify that the **Type** and **System** options are set to your satisfaction.
9. Click **Finish**.

Edit an equipment foundation

1. Click **Select** .
2. Select **Foundations** in the **Locate Filter**.
3. Select the foundation to edit.
4. Using the ribbon, edit the selected foundation as needed.

Edit equipment foundation properties

1. Click **Select** on the vertical toolbar.
2. Select **Equipment Foundations** in the **Locate Filter**.
3. Select the equipment foundation to edit.
4. Click **Edit > Properties**.
5. Edit the properties as needed.

Delete an equipment foundation

1. Click **Select** .
2. Select **Equipment Foundations** in the **Locate Filter**.
3. Select the foundation to delete.
4. Click **Delete** .

Equipment Foundation Properties Dialog Box

Specifies the properties for the equipment foundation that you are editing.

See Also

Definition Tab (Equipment Foundation Properties Dialog Box) (on page 487)
Occurrence Tab (Equipment Foundation Properties Dialog Box) (on page 484)
Relationship Tab (on page 63)
Configuration Tab (on page 64)
Notes Tab (on page 65)

Occurrence Tab (Equipment Foundation Properties Dialog Box)

The **Occurrence** tab displays in a grid the equipment foundation properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one equipment foundation, and then selected the properties command, only the common properties between the selected equipment foundations display.

When viewing properties for a single equipment foundation, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on properties. You can access this document using the **Help > Printable Guides** command in the software.

Category

Select the properties that you want to view for the equipment foundation. Equipment foundation properties are divided into several different categories: Standard, Weight and CG, Fabrication and Construction, and Position and Orientation. You select the category to define values for by using the **Category** option.

Standard

Name

Displays the name of the equipment foundation. The equipment foundation name is based on the **Name Rule** selection. If you want to type a new name for the equipment foundation, in the **Name Rule** box, select **User Defined**, and then type a name for the equipment foundation in the **Name** box.

Name Rule

Specify the naming rule that you want to use to name this equipment foundation.

- **Default Name Rule** - Names the equipment foundation using the format "<equipment foundation type>-<location>-<index>" where <equipment foundation type> is what you select in the **Type** box on the ribbon, <location> is the global workshare location ID, and <index> is a unique index number that starts at 0001. For example, BlockEqpFndn-1-0043.
- **Unique Name Rule** - Names the equipment foundation using the format "<system>-<equipment foundation type>-<location>-<index>" where <system> is the name of the parent system to which the equipment foundation belongs, <equipment foundation type> is what you select in the **Type** box on the ribbon, <location> is the global workshare location ID, and <index> is a unique index number that starts at 0001. For example, Structure System-BlockEqpFndn-1-0043.
- **User Defined** - Select to specify the footing name yourself in the **Name** box.

System

Select the system to which the equipment foundation that you are placing belongs.

Block Material

Select a material for the equipment foundation. Materials are defined in the **AllCommon.xls** workbook or in the Catalog task.

Block Grade

Select a material grade for the equipment foundation. Material grades are defined in the **AllCommon.xls** workbook or in the Catalog task.

Block Length

Specifies the length of the equipment foundation.

Block Width

Specifies the width of the equipment foundation.

Block Height

Specifies the height of the equipment foundation.

Block Sizing By Rule

Set to **True** to allow the software to select the equipment foundation to use based on the default foundation defined for the equipment part. Set to **False** to enter the dimensions of the equipment foundation.

Block Edge Clearance

Specifies the clearance between the equipment and the equipment foundation.

Reporting Requirement

Specify whether this object is reported.

Reporting Type

Select the reporting requirements code for the object. Valid codes are defined in Catalog in the **Reporting Type** select list.

Volume

Displays the volume of the equipment foundation (length × width × height).

Surface Area

Specifies the surface area of the equipment foundation (top surface plus the bottom surface plus the side surfaces × length).

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material density multiplied by the object's solid volume. Therefore, the material of the object affects the weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight

Specifies the dry weight of the object.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Dry WCG Origin

Specifies the way in which the dry weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the dry weight center-of-gravity location relative to the active coordinate system.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Position and Orientation

East

Displays the distance of the connection point from the active coordinate system origin in the east direction. You can change this value when the equipment foundation is placed by point, whereas when placed by equipment, it will be set to read only.

North

Displays the distance of the connection point from the active coordinate system origin in the north direction. You can change this value when the equipment foundation is placed by point, whereas when placed by equipment, it will be set to read only.

Elevation

Displays the distance of the connection point above or below the active coordinate system origin. You can change this value when the equipment foundation is placed by point, whereas when placed by equipment, it will be set to read only.

Bearing

Displays the bearing angle for the shape. This angle sets the reference on the object to the X-axis of its local coordinate system. The local X-axis is the default axis of primary symmetry for all symbols in the catalog. This option sets the reference in the model to the active coordinate system North axis. The bearing measurement direction is clockwise from

the active coordinate system north looking in the negative active coordinate system direction. That is, down from 0 to 360 degrees. You can enter negative bearing angles, but the software will automatically convert them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the bearing measure displays 0 degrees. You can change this value when the equipment foundation is placed by point, whereas when placed by equipment, it will be set to read only.

Pitch

Displays the pitch angle for the shape. This angle sets the reference on the object to the X-axis of its local coordinate system. This option sets the reference in the model to a line that is the intersection of the vertical plane through the X-axis of the local coordinate system and the active coordinate system horizontal plane. The angle is measured in the positive direction from the horizontal plane in the active coordinate system up direction regardless of the current bearing. Pitch angles are limited to between -90 degrees and +90 degrees with 0 indicating horizontal. You can change this value when the equipment foundation is placed by point, whereas when placed by equipment, it will be set to read only.

Roll

Displays the roll angle for the shape. This angle sets the reference on the object to the Y-axis of its local coordinate system. This option sets the reference in the model to a line that is perpendicular to the local coordinate system x axis and in the horizontal plane. Roll angles are measured clockwise from horizontal to the y-axis of the local coordinate system. The roll angle is between 0 and 360 degrees. You can enter negative roll angles, but the software will automatically convert them to the positive equivalents. If the pitch is set to +/- 90 degrees, then the reference in the model is the North axis. You can change this value when the equipment foundation is placed by point, whereas when placed by equipment, it will be set to read only.

See Also

Equipment Foundation Properties Dialog Box (on page 483)

Definition Tab (Equipment Foundation Properties Dialog Box)

The **Definition** tab displays the equipment foundation properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one equipment foundation, and then selected the properties command, only the common properties between the selected equipment foundations display.

The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties. You can access this document using the **Help > Printable Guides** command in the software.

See Also

Equipment Foundation Properties Dialog Box (on page 483)

SECTION 31

Member Autoconnect

Connects existing built-up or standard members using a pre-defined set of rules and tolerances. You can either let the software locate supporting objects, or select them manually. Generally, when the software locates the supporting members, supporting members must be within 25 cm of the end of a supported member. If more than one supporting candidate is found, built-in selection rules decide which supporting member to use considering axis directions and relative position along. Member type and cross-section size is ignored. If you selected the supporting objects, the selection is simply based on the nearest one.

The software operates only on Frame Connections that are Unsupported and have no supporting object. Best results occur by using the Collinear module first, then the Point On module, and then End module for braces. Collinear requires the supporting member to be collinear with the supported one. In automatic mode, this module will search for collinear members successively. If they are within 10 degrees of vertical, locate only the top so that the upper ones use the lower as their supporting.

Member Autoconnect Ribbon

Select Member or Frame Connection to Connect

Select the members, or the frame connection of the members, to connect. These members are the supported members and will be extended or shortened if you select the **Extend Members** option.

Select Target Object

Select the members, grid lines, slabs, or walls to which to connect. These objects are the supporting objects.

Connection method

Select how to search for possible target objects.

- **Collinear** - Search for target objects within 25 cm that are collinear to the selected members. This option places axis-end frame connections. You typically use this option for columns that are stacked on top of each other. This option transverses members successively until all collinear members are connected. This option ignores horizontal and near-horizontal members from consideration allowing it to process vertical braces and columns.
- **Point on** - Search for target objects within 25 cm that are not collinear to the selected members. This option places axis-along frame connections. You can manually select the target supporting object or have the command automatically select the supporting objects. If the member is horizontal and the selected target is vertical and an elevation plane is found, the elevation plane is included in the connection.
- **Surface** - Search for surfaces of slabs, walls, and plates within 25 cm to the selected members. This option places surface-default frame connections.
- **Grid** - Creates a relationship between the selected member and the selected grid lines. The grid line must intersect the end of the member.

- **End** - Search for member ends in the model within 25 cm of the supported member frame connection. This option places axis-end frame connections or vertical-corner-brace connections. You typically use this option for braces.

Filter out connected connections

Click to remove connected frame connections from the selection set. Only unsupported frame connections will remain in the selection set after you click this option.

Accept Selection

Click to confirm that the objects you have selected are the ones to use.

Reject Selection

Click to clear the object selections that you have made.

Finish

Updates the frame connections on the selected members.

Extend Members

Select to have the software automatically extend or shorten the selected members to make the required connection possible.

What do you want to do?

- *Apply axis-end frame connections to stacked columns* (on page 489)
- *Connect beams to columns* (on page 490)
- *Connect horizontal braces and beams to supporting beams* (on page 490)
- *Connect members to slabs, walls, and plates* (on page 491)
- *Connect braces to members* (on page 491)
- *Connect Members to Grids* (on page 492)

Apply axis-end frame connections to stacked columns

1. Click **Member Autoconnect** .
2. Set **Connection Method** to **Collinear**.
3. Select the columns to process. You only have to select the topmost or bottommost column in the stack. This command transverses members successively until all collinear members are connected.
4. Optionally, click **Filter out connected connections** to remove connected frame connections from your selection. Only unsupported frame connections will remain in your selection.
5. Click **Accept** .
6. Click **Finish**.

The command processes the selected frame connections. Frame connections that were modified display in white. Frame connections that were not modified display in red.

7. Click **Yes** to use the unmodified red frame connections as your starting selection set for the next run of the command.

   or

   Click **No** to clear the unmodified red frame connections from your select set.

Connect beams to columns

1. Click **Member Autoconnect** .
2. Set **Connection Method** to **Point on**. This option places axis-along frame connections. You can manually select the target supporting object or have the command automatically select the supporting objects. If the member is horizontal and the selected target is vertical and an elevation plane is found, the elevation plane is included in the connection.
3. Select the members to process.
4. Optionally, click **Filter out connected connections** to remove connected frame connections from your selection. Only unsupported frame connections will remain in your selection.
5. Click **Accept** .
6. Click **Finish**.

   The command processes the selected frame connections. Frame connections that were modified display in white. Frame connections that were not modified display in red.

7. Click **Yes** to use the unmodified red frame connections as your starting selection set for the next run of the command.

   or

   Click **No** to clear the unmodified red frame connections from your select set.

Connect horizontal braces and beams to supporting beams

1. Click **Member Autoconnect** .
2. Set **Connection Method** to **Point on**. This option places axis-along frame connections. You can manually select the target supporting object or have the command automatically select the supporting objects. If the member is horizontal and the selected target is vertical and an elevation plane is found, the elevation plane is included in the connection.
3. Select the members to process.
4. Optionally, click **Filter out connected connections** to remove connected frame connections from your selection. Only unsupported frame connections will remain in your selection.
5. Click **Accept** .

6. Click **Finish**.

   The command processes the selected frame connections. Frame connections that were modified display in white. Frame connections that were not modified display in red.

7. Click **Yes** to use the unmodified red frame connections as your starting selection set for the next run of the command.

   or

   Click **No** to clear the unmodified red frame connections from your select set.

Connect members to slabs, walls, and plates

1. Click **Member Autoconnect**  .
2. Set **Connection Method** to **Surface**. This option places surface-default frame connections.
3. Select the members to process.
4. Optionally, click **Filter out connected connections**  to remove connected frame connections from your selection. Only unsupported frame connections will remain in your selection.
5. Click **Accept**  .
6. Click **Finish**.

   The command processes the selected frame connections. Frame connections that were modified display in white. Frame connections that were not modified display in red.

7. Click **Yes** to use the unmodified red frame connections as your starting selection set for the next run of the command.

   or

   Click **No** to clear the unmodified red frame connections from your select set.

Connect braces to members

1. Click **Member Autoconnect**  .
2. Set **Connection Method** to **End**. This option places axis-end frame connections or vertical-corner-brace connections. You would use this option typically for braces.
3. Select the members to process.
4. Optionally, click **Filter out connected connections**  to remove connected frame connections from your selection. Only unsupported frame connections will remain in your selection.
5. Click **Accept**  .
6. Click **Finish**.

   The command processes the selected frame connections. Frame connections that were modified display in white. Frame connections that were not modified display in red.

7. Click **Yes** to use the unmodified red frame connections as your starting selection set for the next run of the command.

   or

   Click **No** to clear the unmodified red frame connections from your select set.

Connect Members to Grids

1. Click **Member Autoconnect** .
2. Set **Connection Method** to **Grid**. This option creates a relationship between the selected member and the selected grid lines. The grid line must intersect the end of the member.
3. Select the members to process.
4. Click **Accept** .
5. Select the grid line.
6. Click **Accept** .
7. Click **Finish**.

   The command processes the selected frame connections. Frame connections that were modified display in white. Frame connections that were not modified display in red.
8. Click **Yes** to use the unmodified red frame connections as your starting selection set for the next run of the command.

   or

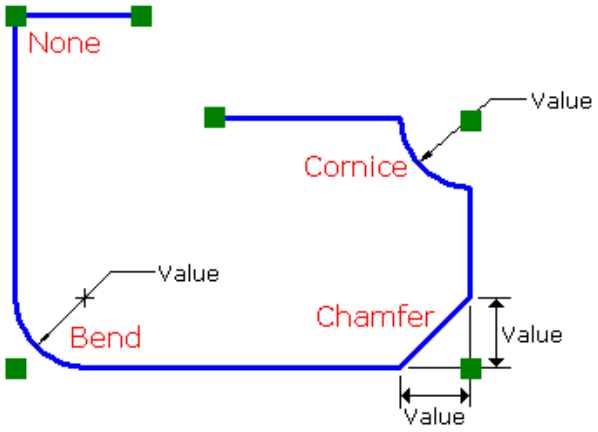
   Click **No** to clear the unmodified red frame connections from your select set.

SECTION 32

Sketching in 3-D

To place handrails, curved members, and one of the methods for placing slabs and walls in the model is to sketch a path in the 3-D environment. For handrails, the sketched 3-D path specifies the handrail location. For curved members, the sketched 3-D path specifies the curved member location. For slabs, the sketched 3-D path specifies the slab boundaries or a portion of the slab boundaries. For walls, the sketched 3-D path specifies the wall location. You can use **Tools > PinPoint**, **Tools > Point Along**, and the SmartSketch relationship indicators when defining the path to help with precise placement.

Path segments can be straight lines or three-point arcs with the ability to switch between the two types at any time using the **Path Type** option. In addition, you can define four types of corners for your path using the **Turn Type** option. The following graphic shows examples of the available turn types (paths display in the model as yellow lines (blue below) with yellow squares (green below) at the vertices):



When placing a path for a handrail or curved member, the path can transverse multiple elevations and not required to be a closed shape. However, when placing a path that defines the boundaries of a slab or the path of a wall, the software limits the path to the plane defined for the slab or wall.

Create Path Ribbon

Sets options for defining a new path. The path can be used to define the location of a handrail, curved member, the boundaries of a slab, or the path of a wall.

Properties

Displays the **Sketch Properties** dialog box, in which you can view properties for the path. For more information, see *Sketch Properties Dialog Box* (on page 506).

Finish

Displays the path in the active view and returns control to the activating command.

Cancel

Cancels the path placement, closes the ribbon, and returns control to the activating command.

Show Cross Section View

Opens a new graphics window that is oriented normal to the sketch plane.

Edit

Allows you to modify and move the existing path. When you initially create a path, this option is only available after you place at least two points in the path. You can select the segment, turn, or multiple segments to which you want to make modifications.

Create

Allows you to sketch the path or add segments to an existing path.

First Point

Specifies that you are currently defining the first point of the path segment.

Second Point (Straight Line)

Specifies that you are currently defining the second point of a straight path segment.

Second Point (Arc)

Specifies that you are currently defining the second point of an arc. This option only appears when **Arc by 3 Points** is selected in the **Path Type** list.

Third Point (Arc)

Specifies that you are defining the final point of an arc.

Insert Vertex

Allows you to add a vertex to a straight segment. After the vertex has been inserted, you can move that vertex to edit the segment path. You cannot insert a vertex on arc path segments.

Path Type

Specifies the type of line for the current segment in the path. To change the segment type, click a new type in the **Path Type** list. You can select from the following line types:

- **Line** - Defines the line type for the segment to be a straight line.
- **Arc by 3 Points** - Defines the line type for the segment to be an arc. To define the arc, you must click three points in the view.
- **No Line** - This option is not available for handrails, walls, or slabs.
- **Arc by End Points** - Defines the line type for the segment to be an arc that you have defined by the arc's end points. To define the arc, you must define the arc's two end points and then a point along the arc.
- **Elliptical Arc** - Defines the line type for the segment to be an elliptical arc.

Plane

Activates options for selecting a working plane for the path. This option is disabled when defining a path for a slab.

- **Plan Plane** - Defines the work surface as the XY plane at the depth of the active end. You also can press 1 to select this option.
- **Elevation Plane: East-West** - Defines the work surface as the XZ plane. You also can press 2 to select this option.
- **Elevation Plane: North-South** - Defines the work surface as the YZ plane. You also can press 3 to select this option.
- **Plane by Turn** - Defines the work surface as the plane defined by an existing turn. You select the turn to set the plane. You also can press 4 to select this option.
- **Plane by Three Points** - Defines the work surface using three points that you define. You also can press 5 to select this option.
- **No Plane** - Clears any work surfaces. The software does not project points that you place to any plane. You also can press 6 to select this option.

Lock Angle

Locks or unlocks the **Angle** box. Locking the corresponding angle value creates a constraint along which the selected turn angle can be moved.

Angle

Specifies the angle for the turn.

Lock Length

Locks or unlocks the Length box.

Length

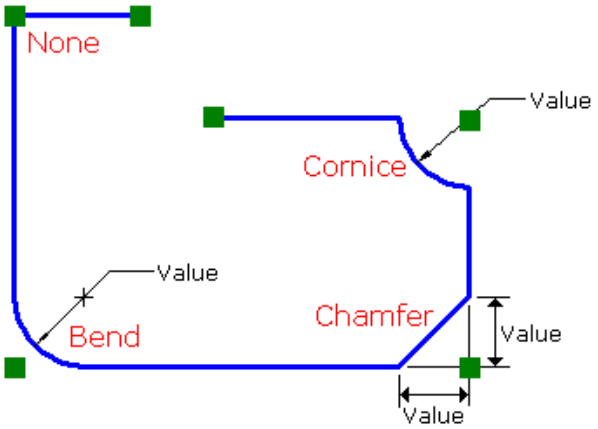
Specifies the length of the selected path segment.

Turn Type

Specifies the type of turn associated with the current path segment. The **Turn Type** option is unavailable if you select **Arc by 3 Points** or **No Line** in the **Path Type** list. The following turn types are available:

- **None** - Indicates that no special turn type will be applied to the turn.
- **Bend** - Specifies that the turn type between two segments is a bend. You can specify the angle of the bend in the **Value** box.
- **Cornice** - Specifies that the turn type between two segments is a cornice. You can specify the radius of the cornice in the **Value** box.
- **Chamfer** - Specifies that the turn type between two segments is a chamfer. You can specify the dimensions for setback value of the chamfer in the **Value** box. The dimensions of setback A and setback B for the chamfer must be the same.
- **Split** - Specifies that you want to split the path you are defining into multiple individual paths between each set of vertices. This option is only available when **Path Type** is set to **Line** and the previous vertex is not the end vertex of an arc. Use this option when, for example, you want individual handrails for each straight segment of the path.

The following graphic includes an example of each of the available turn types:



Value

Defines dimensions for the selected turn type.

Edit Path Straight Segment Ribbon

Sets options for modifying a straight segment of an existing path. This ribbon appears when you select a straight segment in the existing path.

Properties

Displays the **Sketch Properties** dialog box, in which you can view properties for the path.

Finish

Displays the path in the active view and returns control to the **Place Handrail** or **Place Slab** command.

Cancel

Cancels the path placement, closes the ribbon, and returns control the activating command.

Show Cross Section View

Opens a new graphics window that is oriented normal to the sketch plane.

Edit

Allows you to modify and move the existing path. You can select the segment, turn, or multiple segments to which you want to make modifications.

Create

Displays the **Create Path** ribbon bar to allow you to add segments to an existing path. You cannot add segments to slab boundary paths.

First Point

Specifies that you are selecting the starting location of the move vector.

Second Point

Specifies that you are selecting the ending location of the move vector.

Insert Vertex

Allows you to add a vertex to the selected straight segment. After you define the location of the vertex along the segment, click **Finish** to insert the vertex. After the vertex has been inserted, you can move that vertex to edit the segment path. You cannot insert a vertex on arc path segments.

Path Type

Specifies the type of line for the current segment in the path. To change the segment type, click a new type in the **Path Type** list. You can select from the following line types:

- **Line** - Defines the line type for the segment to be a straight line.
- **Arc by 3 Points** - Defines the line type for the segment to be an arc. To define the arc, you must click three points in the view.
- **No Line** - This option is not available for handrails or slabs.
- **Arc by End Points** - Defines the line type for the segment to be an arc that you defined by the arc's end points. To define the arc, you must define the arc's two end points and then a point along the arc.
- **Elliptical Arc** - Defines the line type for the segment to be an elliptical arc

Plane

Activates options for selecting a working plane for the path. This option is disabled if you are editing a path for a slab boundary. Six options are available:

- **Plan Plane** - Defines the work surface as the XY plane at the depth of the active end. You also can press 1 to select this option.
- **Elevation Plane: East-West** - Defines the work surface as the XZ plane. You also can press 2 to select this option.
- **Elevation Plane: North-South** - Defines the work surface as the YZ plane. You also can press 3 to select this option.
- **Plane by Turn** - Defines the work surface as the plane defined by an existing turn. You select the turn to set the plane. You also can press 4 to select this option.
- **Plane by Three Points** - Defines the work surface using three points that you define. You also can press 5 to select this option.
- **No Plane** - Clears any work surfaces. The software does not project points that you place to any plane. You also can press 6 to select this option.

Delete Selected Items

Deletes the selected path segments.

Length Locked

Defines whether or not the length of the selected segment should remain constant while moving.

When locked  , the software automatically modifies the turn points, along with the length and angle of adjacent segment, to remain connected to the moved segment. The length of the moved segment does not change.

When not locked  , the software extends or shortens the associated segments to connect

with the new position of the moved segment. The length of the moved segment can change.

Edit Path Arc Ribbon

Sets options for modifying an arc that is part of an existing path. This ribbon appears when you select an arc in the existing path.

Properties

Displays the **Sketch Properties** dialog box, in which you can view properties for the path.

Finish

Displays the path in the active view and returns control to the **Place Handrail** or **Place Slab** command.

Cancel

Cancels the path placement, closes the ribbon, and returns control the activating command.

Show Cross Section View

Opens a new graphics window that is oriented normal to the sketch plane.

Edit

Allows you to modify and move the existing path. You can select the segment, turn, or multiple segments to which you want to make modifications.

Create

Displays the **Create Path** ribbon bar to allow you to add segments to an existing path. You cannot add segments to slab boundary paths.

First Point

Specifies that you are selecting the starting location of the move vector.

Second Point

Specifies that you are selecting the ending location of the move vector.

Path Type

Specifies the type of line for the current segment in the path. To change the segment type, click a new type in the **Path Type** list. You can select from the following line types:

- **Line** - Defines the line type for the segment to be a straight line.
- **Arc by 3 Points** - Defines the line type for the segment to be an arc. To define the arc, you must click three points in the view.
- **No Line** - This option is not available for handrails.

Plane

Activates options for selecting a working plane for the path. This option is disabled if you are editing a path for slab boundaries. Six options are available:

- **Plan Plane** - Defines the work surface as the XY plane at the depth of the active end. You also can press 1 to select this option.
- **Elevation Plane: East-West** - Defines the work surface as the XZ plane. You also can press 2 to select this option.

- **Elevation Plane: North-South** - Defines the work surface as the YZ plane. You also can press 3 to select this option.
- **Plane by Turn** - Defines the work surface as the plane defined by an existing turn. You select the turn to set the plane. You also can press 4 to select this option.
- **Plane by Three Points** - Defines the work surface using three points that you define. You also can press 5 to select this option.
- **No Plane** - Clears any work surfaces. The software does not project points that you place to any plane. You also can press 6 to select this option.

Delete Selected Items

Deletes the selected path segments.

Edit Path Turn Ribbon

Sets options for modifying a turn in an existing path. This ribbon appears when you select a turn in the existing path.

Properties

Displays the **Sketch Properties** dialog box, in which you can view properties for the path.

Finish

Displays the path in the active view and returns control to the **Place Handrail** or **Place Slab** command.

Cancel

Cancels the path placement, closes the ribbon, and returns control the activating command.

Show Cross Section View

Opens a new graphics window that is oriented normal to the sketch plane.

Edit

Allows you to modify and move the existing path. You can select the segment, turn, or multiple segments to which you want to make modifications.

Create

Displays the **Create Path** ribbon bar to allow you to add segments to an existing path. You cannot add segments to slab boundary paths.

First Point

Specifies that you are selecting the starting location of the move vector.

Second Point

Specifies that you are selecting the ending location of the move vector.

Plane

Activates options for selecting a working plane for the path. This option is disabled if you are editing a path for a slab boundary. Six options are available:

- **Plan Plane** - Defines the work surface as the XY plane at the depth of the active end. You also can press 1 to select this option.

- **Elevation Plane: East-West** - Defines the work surface as the XZ plane. You also can press 2 to select this option.
- **Elevation Plane: North-South** - Defines the work surface as the YZ plane. You also can press 3 to select this option.
- **Plane by Turn** - Defines the work surface as the plane defined by an existing turn. You select the turn to set the plane. You also can press 4 to select this option.
- **Plane by Three Points** - Defines the work surface using three points that you define. You also can press 5 to select this option.
- **No Plane** - Clears any work surfaces. The software does not project points that you place to any plane. You also can press 6 to select this option.

Delete Selected Items

Deletes the selected path segments.

Angle 1

Displays the first angle used in the turn, if one exits. This option is read-only.

Angle 2

Displays the second angle used in the turn, if one exits. This option is read-only.

Angle 3

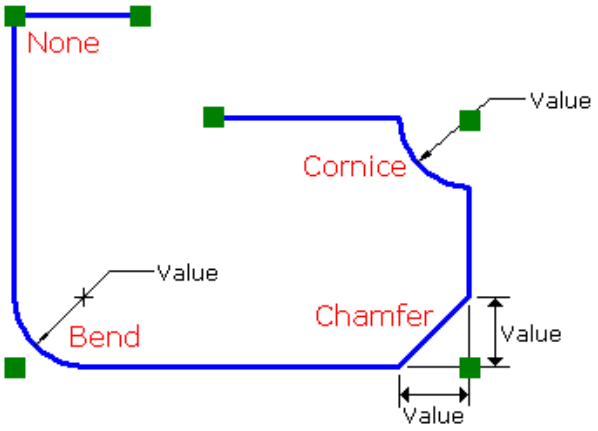
Displays the third angle used in the turn, if one exits. This option is read-only.

Turn Type

Specifies the type of turn. You can change the turn type by selecting another type in the list. The following turn types are available:

- **None** - Indicates that no special turn type will be applied to the turn.
- **Bend** - Specifies that the turn type between two segments is a bend. You can specify the angle of the bend in the **Value** box.
- **Chamfer** - Specifies that the turn type between two segments is a chamfer. You can specify the dimensions for the chamfer in the **Value** box. The dimensions of setback A and setback B for the chamfer must be the same.
- **Cornice** - Specifies that the turn type between two segments is a cornice. You can specify the radius of the cornice in the **Value** box.

The following graphic includes an example of each of the available turn types:



Value

Defines dimensions for the selected turn type.

What do you want to do?

- *Define a path* (on page 501)
- *Add segments to a path* (on page 502)
- *Add vertices to a path* (on page 503)
- *Move segments of a path* (on page 503)
- *Modify a straight segment in a path* (on page 504)
- *Modify an arc in a path* (on page 505)
- *Modify a turn in a path* (on page 505)

Define a path

1. Click to place the first point for the path.

   TIPS

   - You can use Pin Point, Point Along, and the SmartSketch relationship indicators when defining your path.
   - For handrails, you can change the path plane at any time by selecting a new plane in the **Plane** list.
   - To change the segment from a straight line to an arc, click **Arc by 3 Points** in the **Path Type** list, and then click 3 points to define the arc.
   - To switch back to a straight line after sketching an arc, click **Line** in the **Path Type** list.

2. Click the second point for the path.

   TIPS

   - To change turn type for the corner, click the new turn type in the **Turn Type** list and define the angle or offset dimensions for the turn in the **Value** box
   - Use the **Split** turn type to place individual walls or handrails for each straight segment of the path.

3. Click to place other segments of the path as needed.
4. After you place all the points that define the path, click **Finish**.

NOTES

- The software does not require that you close the path for a handrail. You can end the path wherever you want.
- The software does require a closed shape for a slab boundary. If you do not define a closed shape path, then you need to specify additional objects in the model for the missing boundary sides.
- After you place a segment of the path by defining two points, you can click **Edit** on the ribbon to change the segment.
- To manually set the length and angle for a segment of the path, change the values in the **Angle** and **Length** boxes on the ribbon.

Add segments to a path

1. Click **Select** .
2. Select the handrail for which you want to modify the path.
3. On the ribbon, click **Path** .
4. To add one or more segments to the path, click **Create**.
5. Click the point on the existing path where you want to insert the new segment.

   TIPS

   - You can use Pin Point, Point Along, and the SmartSketch relationship indicators when defining your path.
   - You can change the plane for the path at any time by selecting the plane in the **Plane** list.
   - To change the segment from a straight line to an arc, click **Arc by 3 Points** in the **Path Type** list, and then click 3 points to define the arc.
   - To switch back to a straight line after sketching an arc, click **Line** in the **Path Type** list.

6. Click to place other points and add to the path as needed.

   TIP

   - To change turn type for the corner, click the new turn type in the **Turn Type** list and define the angle or dimensions for the turn in the **Feature Value** box.

7. After you place all the points for the new segments, click **Finish**.

NOTES

- You cannot add segments to slab boundary paths.
- You can click **Edit** on the ribbon to change the segment or modify the path further.
- To manually set the length and angle for a segment of the path, change the values in the **Angle** and **Length** boxes on the ribbon.

Add vertices to a path

1. Click **Select** .
2. Select the handrail for which you want to modify the path.
3. On the ribbon, click **Path** .
4. Select the segment to add the vertex to.
5. On the ribbon, click **Insert Vertex** .
6. Identify the location on the path segment for the new vertex.
7. Click **Finish**.

Move segments of a path

1. Click **Select** .
2. Select **Structure** in the **Locate Filter**.
3. Select the handrail or slab for which you want to modify the path.
4. For handrails, click **Path** on the ribbon.
   -OR-
   For slabs, click **Define Boundaries** and then click **Sketch 3D** .
5. Select the segments to move.
6. If you want the length of a straight segment to remain constant while you move the segment click **Length Locked** .

   TIPS

   - When the segment length is locked, the software automatically modifies the turn points, along with the length and angle of adjacent segments, to remain connected to the moved segment. The length of the moved segment does not change.
   - When the segment length is not locked, the software extends or shortens the associated segments to connect with the new position of the moved segment. The length of the moved segment can change.
7. Click to specify the starting location of the move vector.
8. Click to specify the ending location of the move vector.

9. Click **Finish**.

NOTES

- The new path displays in dynamics when you click **Finish**.
- The new handrail or slab is actually created when you click **Accept** and then **Finish** on the ribbon.

Modify a straight segment in a path

1. Click **Select**.
2. Select **Structure** in the **Locate Filter**.
3. Select the handrail or slab for which to modify the path.
4. For handrails, click **Path** on the ribbon.
   -OR-
   For slabs, click **Define Boundaries** and then click **Sketch 3D**.
5. Select the straight segment to modify.

   TIP You can select multiple segments by holding down the **Ctrl** key and clicking the segments.
6. To change the line type for the segment, click a new type in the **Path Type** list.
7. To change the plane for the segment, click a new plane in the **Plane** list.
8. To delete the segment, click **Delete**.
9. If you want the length of a straight segment to remain constant when you move the segment click **Length Locked**.

   TIPS

   - When the segment length is locked, the software automatically modifies the turn points, along with the length and angle of adjacent segment, to remain connected to the moved segment. The length of the moved segment does not change.
   - When the segment length is not locked, the software extends or shortens the associated segments to connect with the new position of the moved segment. The length of the moved segment can change.
10. Click **Finish**.

NOTES

- The new path displays in dynamics when you click **Finish**.
- The new handrail or slab is actually created when you click **Accept** and then **Finish** on the ribbon.
- You can also move the segments of a path. For more information, see *Move Segments of a Path* (on page 503).

Modify an arc in a path

1. Click **Select** .
2. Select **Structure** in the **Locate Filter**.
3. Select the handrail or slab for which to modify the path.
4. For handrails, click **Path** on the ribbon.
   -OR-
   For slabs, click **Define Boundaries** and then click **Sketch 3D** .
5. Select the arc to modify.

   TIP You can select multiple segments by holding down the **Ctrl** key and clicking the segments.
6. To change the line type for the segment, click a new type in the **Path Type** list.
7. To change the plane for the segment, click a new plane in the **Plane** list.
8. To delete the segment, click **Delete** .
9. Click **Finish**.

NOTES

- The new path displays in dynamics when you click **Finish**.
- The new handrail or slab is actually created when you click **Accept** and then **Finish** on the ribbon.
- You can also move the segments of a path. For more information, see *Move Segments of a Path* (on page 503).

Modify a turn in a path

1. Click **Select** .
2. Select **Structure** in the **Locate Filter**.
3. Select the handrail or slab for which to modify the path.
4. For handrails, click **Path** on the ribbon.
   -OR-
   For slabs, click **Define Boundaries** and then click **Sketch 3D** .
5. To change the plane for the segment, click a new plane in the **Plane** list.
6. To delete the turn, click **Delete** .
7. To change the turn type, select a new type in the **Turn Type** list.
8. Click **Finish**.

NOTES

- The new path displays in dynamics when you click **Finish**.

- The new handrail or slab is actually created when you click **Accept** ✓ and then **Finish** on the ribbon.
- You can also move the segments of a path. For more information, see *Move Segments of a Path* (on page 503).

Sketch Properties Dialog Box

Sets options for paths. You cannot edit the different applicable properties.

General Tab (Sketch Properties Dialog Box) (on page 506)
Configuration Tab (on page 64)
Relationship Tab (on page 63)
Notes Tab (on page 65)

General Tab (Sketch Properties Dialog Box)

Displays the points that make up a path, their X, Y, and Z coordinates, turn types, and turn type dimensions.

Point No

Displays point number that identifies the selected point.

X

Displays the location of the point on the X-axis.

Y

Displays the location of the point on the Y-axis.

Z

Displays the location of the point on the Z-axis.

Turn Type

Displays the type of turn associated with the point. Turn types include none, bend, chamfer, and cornice.

Value

Specifies dimensions for the selected turn type. For bends and cornices, the value specifies the radius of the bend. For chamfers, the value specifies the dimensions for setback A and setback B of the chamfer.

See Also

Sketch Properties Dialog Box (on page 506)

Importing and Exporting Structure

Structural objects are imported and exported from Smart 3D using the CIMsteel Integration Standards 2.0 (CIS/2). You can only export concrete slabs, walls, structural systems, openings on slabs and walls, footings, equipment foundations, and design equipment (with an equipment classification of civil elements or structural elements) to the Industry Foundation Class (IFC) format. Steel members can be exported only through the CIS/2 format. You can also export objects to IGES, ACIS, and neutral XML formats using **File > Export** commands available when in the **Molded Forms** or **Structural Detailing** tasks.

Object	CIS/2 Export	CIS/2 Import	IFC Export	bocad Import
Grids	Yes	No	No	Yes
Linear Member	Yes	Yes	No	Yes
Curved Member	Yes	Yes	No	No
Built Up Member	Yes	Yes	No	No
Slab Object	Yes	Yes	Yes	No
Wall Object	Yes	Yes	Yes	No
Fireproofing	No	No	No	No
Traffic Items (stairs, ladders, handrails)	Yes	Yes[1]	No	Yes[1]
Footing Object	No	No	Yes	No
Equipment Foundation Object	No	No	Yes	No
Designed Solid - Structural	No	No	Yes	No
Openings	Yes	Yes	Yes	Yes
Marine Planar Plate	Yes	Yes	No	Yes
Marine Linear Profile	Yes	No	No	No
1 - A parent item with parts				

Importing and Undo

Because of the system resources required to support **Undo**, it is possible for you to import a file where you cannot undo the import. The ability to undo is driven by the number objects in each object type being imported. The undo limit can be calculated with this formula:

```
roundUp(members/100) + roundUp(slabs/100) + … + roundUp(stairs/100) ≤
10
```

where:

`members, slabs, stairs,` and so forth are the number of objects in the import file segregated into these object types:

- member
- slab
- opening
- ladder
- ladder member part
- ladder planar part
- handrail planar part
- stair
- stair member part
- stair planar part
- structure system
- structure system member part
- structure system planar part

`roundUp` - Round up the result of (members / 100) to the next whole number. For example, if you have 45 members in the import file, 45 / 100 = .45, so members count as 1. However, if you have 367 members in the import file, 367 / 100 = 3.67 so members count as 4.

The sum total of all object types must be 10 or less for **Undo** to be available.

Example: Your import file has 432 members, 25 slabs, 15 ladders, 69 openings.

```
roundUp(432/100) + roundUp(25/100) + roundUp(15/100) + roundUp(69/100)
roundUp(4.32) + roundUp (.25) + roundUp (.15) + roundUp(.69)
5 + 1 + 1 + 1 = 8
```

8 is less than 10 so you will be able to Undo your import if needed.

Industry Foundation Class

Industry Foundation Class (IFC) was developed by buildingSMART (International Alliance for Interoperability, IAI) to facilitate interoperability in the building industry. For more information about IFC, go to http://www.buildingsmart.com.

Smart 3D provides limited support for IFC by exporting only concrete slabs, walls, structural systems, openings on slabs and walls, footings, equipment foundations, and design equipment (with an equipment classification of civil elements or structural elements) to the Industry Foundation Class (IFC) format. Linear and curved concrete members cannot be exported to IFC. We recommend the CIS/2 format for transferring linear and curved members.

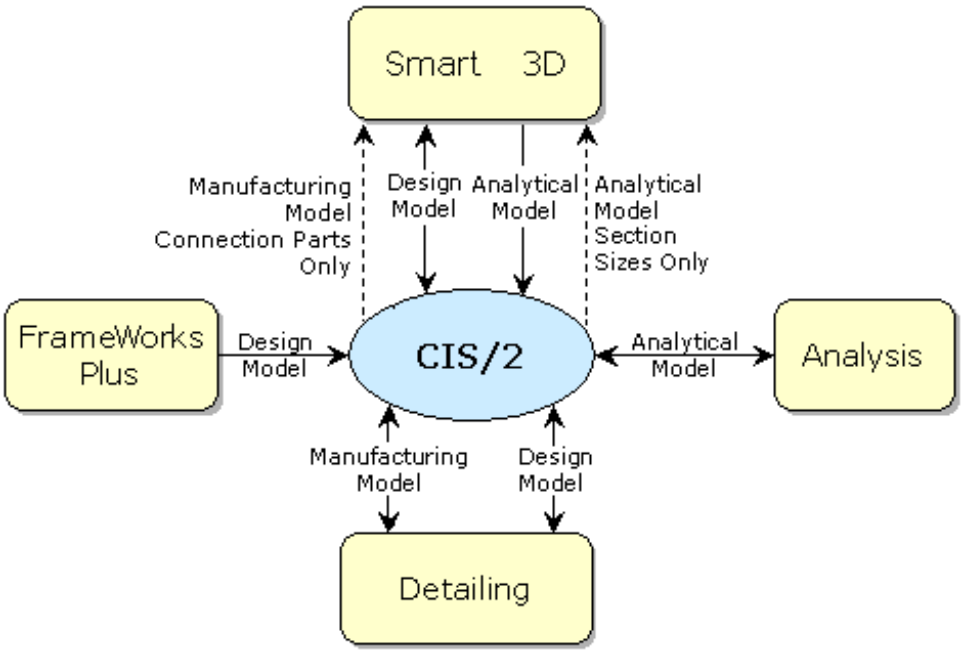
CIS/2 Import and Export

CIS/2 was authored by The Steel Construction Institute and the Computer Aided Engineering Group School of Civil Engineering at the University of Leeds to facilitate a more integrated method of working through the sharing and management of information within and between companies involved in the planning, design, analysis, and construction of steel framed building and similar structures.

Smart 3D provides limited support for:

- importing of manufacturing models

- partial importing and exporting the analytical model (available only in the **Structural Analysis** task)
- importing and exporting the design model



CIS/2 Limitations

There are some limitations when importing and exporting the design model and analysis model through CIS/2:

- Members imported from a design model CIS/2 file are standard or designed Smart 3D members depending on the cross-section specified in the CIS/2 file. Frame connections are created during import and are set to Axis-Along with X, Y, and Z offsets defined if the connection information is provided in the CIS/2 import file. Assembly connections are created during import if the connection information is provided in the CIS/2 import file.
- According to the LPM/6 schema, both the AXIS2_PLACEMENT_3D and its associated location (CARTESIAN_POINT) are GEOMETRIC_REPRESENTATION_ITEM entities. As such, either or both can be assigned a unit through the REPRESENTATION entity. Smart 3D first attempts to retrieve the units assigned to the AXIS2_PLACEMENT_3D entity. If none are found, Smart 3D uses the units assigned to its associated CARTESIAN_POINT entity. If units are not assigned to either entity, then the units are considered to be in standard MKSAR (meters) units. If units are assigned to the AXIS2_PLACEMENT_3D entity, any units assigned to CARTESIAN_POINT are not used.
- When importing members from a CIS/2 file exported from FrameWorks Plus,® tapered members are ignored. Members with workpoint offsets are read in at the physical location of the workpoint offset, but no analytical information is retained. In addition, all member end releases are set to User defined - Free. However, the FrameWorks Plus member name is assigned to the Smart 3D member, and the rotation and reflection settings of the member are maintained.
- FrameWorks Plus solids are ignored. FrameWorks Plus slabs are imported with bottom face orientation regardless of the face setting in FrameWorks Plus.

- Only member section size changes can be imported from an analytical model CIS/2 file.
- The software exports only the operation geometry for handrails, stairs, and ladders to the design model.
- The path of curved members that are exported and then re-imported lose any constraints that may have been applied to the original path.
- Slabs are always read into the model using a sketch 3-D path. You can edit the slab path after the import, if needed. No grid lines associated with the slab are imported, and there are no constraints assigned to the slab.

  The assembly_design_structural_member_planar entity is used to represent a slab or plate if it's planar_member_type is "plate" (and Smart 3D is installed); otherwise, it is imported as a slab. Plate Systems are bounded by a single, continuous and closed 2-D curve
- Designed members (members composed of individual plates) are not exported to CIS/2 using the individual plates. However, the designed members are exported to CIS/2 as standard members of the same size.

Based-on CIS/2 Statement for Import

Application Name:	Intergraph Smart™ 3D
Application Version: Version 2016 (11.0)	**Date:** Monday, September 12, 2016
Translator Version: Version 2016 (11.0)	**Date:** Monday, September 12, 2016
Software Vendor:	Intergraph Process, Power & Marine 300 Intergraph Way Madison, Alabama 35758 U.S.A.

The translators for this application have been implemented in accordance with the second release of the CIMsteel Integration Standards (CIS/2.0) for the following (combination of) Conformance Classes:

- CC312+CC110+CC118+CC249, CC312+CC111+CC118+CC259, CC278

Type of CIS Translator:	**Basic** | DMC | IDI | PMR-enabled
Data exchange capabilities:	**Import** | Export | Import & Export
Level of implementation:	**File Exchange** | In memory | DBMS | KBS
Flavors supported:	EU | **US** | UK
Unit Systems supported:	**SI** | US Imperial

The vendor places the following riders on the operation of the translators:

- No riders.

Date of Statement:	Monday, September 12, 2016
Statement made by:	Intergraph Process, Power & Marine

Based-on CIS/2 Statement for Export

Application Name:	Intergraph Smart™ 3D
Application Version: Version 2016 (11.0)	**Date:** Monday, September 12, 2016
Translator Version: Version 2016 (11.0)	**Date:** Monday, September 12, 2016
Software Vendor:	Intergraph Process, Power & Marine 300 Intergraph Way Madison, Alabama 35758 U.S.A.

The translators for this application have been implemented in accordance with the second release of the CIMsteel Integration Standards (CIS/2.0) for the following (combination of) Conformance Classes:

- CC312, CC110, CC118, CC255, CC331, CC100, CC308

Type of CIS Translator:	Basic | **DMC** | IDI | PMR-enabled
Data exchange capabilities:	Import | **Export** | Import & Export
Level of implementation:	**File Exchange** | In memory | DBMS | KBS
Flavors supported:	EU | **US** | UK
Unit Systems supported:	**SI** | US Imperial

The vendor places the following riders on the operation of the translators:

- managed_data_deleted, managed_data_creation, and managed_data_transaction are not exported.

Date of Statement:	Monday, September 12, 2016
Statement made by:	Intergraph Process, Power & Marine

If you have any questions about using this translator, please contact Intergraph support. You can find support information on our web site: http://support.intergraph.com.

Be sure to check the Compatibility Matrix before you import or export data. Open *https://smartsupport.intergraph.com https://smartsupport.intergraph.com*, select **View Downloads** at the top, and then select **Product Compatibility** under **Useful Links** on the right.

See Also

Import Structure (on page 516)
Import bocad (on page 526)
Export Structure (on page 512)

Export Structure

The **File > Export > Structure** command exports the structural physical model to a CIS/2 or IFC file. For more information, see *Importing and Exporting Structure* (on page 507).

Exporting User-defined Properties

You can map user-defined Smart 3D properties to third-party software attributes and export those properties/attributes to the CIS/2 file using the UserAttribute_Map.xml file. A sample UserAttribute_Map.xml file is delivered with Smart 3D in the *[Product Folder]*\SmartPlantStructure\Symbols\StructureImportExportMaps folder.

Export Structure Model

1. Click **File > Export > Structure**.
2. Select the filter to use to select the objects to export.
3. Type a description.
4. Type your name in the **Author** box and your company name in the **Organization** box.
5. Specify the file name and folder for the export file.
6. Define a mapping file, if needed. For more information on mapping files, see *New Mapping File* (on page 535).
7. Define a log file name and folder.
8. Click **OK**.

NOTES

- You must create the filter before you can export the physical model.
- You can create a mapping file using the **File > New Mapping File Command** that can be used when exporting the physical model.
- You can also use the delivered mapping files located in the *[Product Folder]*\SmartPlantStructure\Symbols\StructureImportExportMaps folder.

Export Structure Dialog Box

Controls how objects are written to the export file. You can export to the CIMsteel Integration Standards 2.0 (CIS/2) or to the Industry Foundation Class (IFC) file format.

NOTES

- You can export only concrete objects such as slabs, walls, structural systems, openings on slabs and walls, footings, equipment foundations, and design equipment (with an equipment classification of civil elements or structural elements) to the IFC format.

- Linear and curved concrete members cannot be exported to IFC. We recommend the CIS/2 format for transferring linear and curved members.
- The International System of Units (SI) is used in the exported CIS/2 file regardless of the settings on the **Tools > Options > Units of Measure** tab.

If you have any questions about using this translator, please contact Intergraph Support. You can find support information on our web site: http://support.intergraph.com.

Be sure to check the Compatibility Matrix before you export data. Open *https://smartsupport.intergraph.com https://smartsupport.intergraph.com*, select **View Downloads** at the top, and then select **Product Compatibility** under **Useful Links** on the right.

Export Standard

Select the industry standard to export.

IFC Options

Defines data, display, and organizational information for the IFC file. For more information, see *Export Options Dialog Box* (on page 514).

Filter

Select the filter to use to identify the model objects to export.

Description

Type a description of the model. This description is included in the export file.

Author

Specifies the person who created the export file. The default is the current system user name.

Organization

Specifies your company or organization name to use in the export file.

Export file

Define the file name and folder path of the export file to write. CIS/2 files have a .stp extension while IFC files have an .ifc extension.

Include mapping file

Select this option to use a mapping file when exporting the members to the CIS/2 file. You use a mapping file to:

- Swap the third-party software name for a section (for example, L3.5X2.5X1/4) with the Smart 3D name for a section (for example, L3-1/2X2-1/2X1/4). You must create the section mapping file using the **File > New Mapping File Command** before you can use the mapping file in this command. Sample mapping files for the FrameWorks Plus AISC table are delivered with the software in the *[Product Folder]*\SmartPlantStructure\Symbols\StructureImportExportMaps folder. For more information, see *New Mapping File* (on page 535).
- Swap the third-party material and material grade name with the Smart 3D name. A sample material mapping file for AISC is delivered with the software in the *[Product Folder]*\SmartPlantStructure\Symbols\StructureImportExportMaps folder.
- Specify the user-attributes that need to be exported and to swap the Smart 3D property names with the third-party user-attributes (properties) names. A sample user attribute

mapping file is delivered with the software in the *[Product Folder]*\SmartPlantStructure\Symbols\StructureImportExportMaps folder.

Mapping file

Specify the mapping file to use if **Include mapping file** option is selected.

Log file

Specify a log file name. You can view the log file by clicking **View Log**.

Submit Job

Activates the **Schedule [Task]** dialog box, which is used to define the batch export of files using SmartPlant Batch Services. For more information on the batch settings, see *Schedule [Task] Dialog Box* (on page 524).

View Log

Displays the export log file. You must click **Apply** when exporting to view the log at the end of processing. If you click **OK**, the dialog box is closed at the end of processing, and you are not able to click **View Log**.

Export Options Dialog Box

Defines properties for the IFC file and the data contained in the export file.

Data Tab (IFC Export Options Dialog Box) (on page 514)
Presentation Tab (IFC Export Options Dialog Box) (on page 515)
Organization Tab (IFC Export Options Dialog Box) (on page 515)

Data Tab (IFC Export Options Dialog Box)

Specifies the properties for the format and accuracy of the data contained in the export file.

Boundary Representation (Brep)

Objects are represented by a series of faces that the software stitches together. For example, a design solid is exported as one object with a series of faces.

Constructive Solid Geometry (CSG)

Objects are represented by a series of solids (cylinders, cubes, and so forth) that the software constructs to create the entire solid. This option only applies to design solids. All other solids (slabs, walls, footings, and so forth) are always written out in boundary representation format.

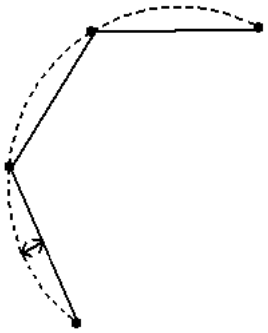
Include Object's Range

Select to include the overall range of an entity in addition to the geometry of each of their components. In general, the entities do not have representation (footings and foundations).

Tessellation or Stroking Tolerance

Specifies the accuracy to use when tessellating nonplanar surfaces or stroking nonlinear

curves.



Presentation Tab (IFC Export Options Dialog Box)

These properties specify how exported objects should render in other software.

Slab

Select a color for slabs.

Wall

Select a color for walls.

Footing

Select a color for footings.

Foundation

Select a color for foundations.

Design Solid

Select a color for foundations.

Organization Tab (IFC Export Options Dialog Box)

These properties give a brief description about the organization that owns the data and is used in the IfcOrganization element.

Name

Type the name of the organization.

Role

Select the role played by the organization.

Description

Type a brief description of the organization.

Role Description

Give a detailed description about the organization and the role it plays. If **Role** is set to **User-defined**, then this field is required.

Address

Specify the address information of the organization.

Import Structure

The **File > Import > Structure > CIS/2** command imports a CIS/2 file into the model. This command recognizes Global User Identities (GUIDs) to uniquely identify objects and manages the electronic exchange with the other software package.

Members imported by this command are either standard or designed members depending on the cross-section specified in the import file. Assembly connections are created if that information is provided in the CIS/2 file and the appropriate import option is selected. Frame connections are always created and connect the members in the import file if the connection information is specified. Otherwise, the frame connections are set to "Unsupported".

If you have any questions about using this translator, please contact Intergraph Support. You can find support information on our web site: http://support.intergraph.com.

Import of Attributes/Properties

User-defined attributes that were defined in the third-party software and exported from that application to the CIS/2 file can be mapped and imported to Smart 3D properties using the UserAttribute_Map.xml file. A sample UserAttribute_Map.xml file is delivered with Smart 3D in the *[Product Folder]*\SmartPlantStructure\Symbols\StructureImportExportMaps folder.

If the third-party attribute that you want to map does not have an equivalent property in Smart 3D, you need to add that property to Smart 3D (using bulkload) before you can map to it in the xml file.

If the third-party attribute does not have an entry in the mapping files but a property with the same name exists in the Smart 3D, then that third-party attribute will still import and populate the Smart 3D property even though it is not in the mapping file.

What do you want to do?

- *Import Structure Model* (on page 516)
- *Preview a CIS file* (on page 517)

Import structure model

1. Click **Tasks > Structure**.
2. Click **File > Import > Structure > CIS/2**.
3. Select the system in which to place the imported objects. You can create new systems in the Systems and Specifications task.
4. Select the coordinate system to which to associate the imported objects. You can create new coordinate systems in the Grids task.
5. Specify which objects to import by selecting the **New** box or which items to update by selecting the **Modified** box.
6. Click **Defaults** and define the default properties for imported objects that do not have recognized properties.

7. Specify the file name and folder for the CIS file.
8. Define a mapping file, if needed.
9. Define a log file name and folder.
10. In the **Action** box, select **Import**.
11. In the **Option** box, select whether the software should use default properties.
12. Click **Apply** to import the CIS file.
13. Click **View Log** to review the log file.

NOTE You can use the **File > New Mapping File** command to create a section name mapping file to use when importing a structural model, or you can use one of the delivered mapping files in the *[Product Folder]*\SmartPlantStructure\Symbols\StructureImportExportMaps folder.

Preview a CIS File

1. Click **Tasks > Structure**.
2. Click **File > Import > Structure > CIS/2**.
3. Select the system in which to place the previewed objects. You can create new systems in the Systems and Specifications task.
4. Select the coordinate system to which to associate the previewed objects. You can create new coordinate systems in the Grids task.
5. Specify which objects to preview by selecting corresponding **New** or **Modified** boxes.
6. Click **Defaults** and define the default properties for imported objects that do not have recognized properties.
7. Specify the file name and folder for the CIS file.
8. Define a mapping file, if needed.
9. Define a log file name and folder.
10. In the **Action** box, select **Preview**.
11. In the **Filter** box, select the preview option to use.
12. Click **Apply** to preview the CIS file.
13. Click **View Log** to review the log file.

NOTE You must create a mapping file using the **File > New Mapping File Command** before you can use that mapping file when importing a structural model.

Import Structure Dialog Box

Controls how objects are imported from the CIS/2 import file. Be sure to check the Compatibility Matrix before you import data. Open *https://smartsupport.intergraph.com* *https://smartsupport.intergraph.com*, select **View Downloads** at the top, and then select **Product Compatibility** under **Useful Links** on the right.

System

Select the system to which to assign the objects being read. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see "Select System Dialog Box" in *Place Linear Member Systems* (on page 35).

Coordinate System

Select the coordinate system to which to assign the objects being read. If needed, you can create a new coordinate system in the Grids task.

New

Loads any object in the CIS/2 file that is not in the model. Use the **All On** or **All Off** commands to select or clear all check boxes.

Modified

Loads any object that exists in both the CIS/2 file and the model that has changed in the CIS/2 file. Use the **All On** or **All Off** commands to select or clear all check boxes.

Members

Select to read into the model the members defined in the CIS/2 file. Openings on members are not imported. Members defined with a built-up cross-section are imported as designed members with these limitations:

- The member must be linear.
- The material and dimensions are defined by the build-up cross-section defined in the catalog. Any material or dimension values specified in the import file are ignored.
- The designed member parts must be imported along with the parent member system. Stand-alone designed member parts (such as connection parts and clip angles) are not imported.

Slabs/Plates

Select to read into the model the slabs and plates defined in the CIS/2 file. Slabs are always read into the model using a sketch 3-D path. You can edit the slab path after the import, if needed. No grid lines associated with the slab are imported, and there are no constraints assigned to the slab.

Walls

Select to read into the model the walls defined in the CIS/2 file. All walls are imported as slabs.

Openings

Select to read into the model the openings (holes) defined in the CIS/2 file for slabs, walls, grating, and checker plates.

Connections

Select to read into the model the gusset plates, clip angles, and other plates and standard sections used to connect members. The detailing software marks these objects as connection parts in the CIS/2 schema during export.

Assemblies

Select to read into the model the assemblies defined in the CIS/2 file.

All On

Click to activate all **New** and **Modified** check boxes.

All Off

Click to clear all **New** and **Modified** check boxes.

Defaults

Activates a dialog box where you define properties to use when the object being imported does not have a recognized property.

Default Properties Dialog Box

The **Default Properties** dialog box defines properties to use when the object being imported does not have a recognized property. You activate this dialog box from the main **Import Structure** dialog box by clicking the **Defaults** button.

Member Defaults

Type Category

Select the type category to use when a member being imported does not have a recognized type category. You can define a custom member type category on the **Structural Member Type** sheet in the **AllCodeLists.xls** workbook.

Type

Select the type to use when a member being imported does not have a recognized type. The properties change depending on the member type that you select. You can define a custom member type on the **Structural Member Type** sheet in the **AllCodeLists.xls** workbook.

Section Name

Select the cross-section to use when a member being imported does not have a recognized section. Sections are defined in the reference data. See the Structure Reference Data Guide for more information about reference data.

Material Name

Select the material to use when a member being imported does not have a recognized material.

Material Grade

Select the material grade to use when a member being imported does not have a recognized material grade.

Slab Defaults

Type

Select the slab type to use when a slab being imported does not have a recognized slab type.

Composition

Select the slab composition to use when a slab being imported does not have a recognized slab composition.

Material Name

Select the material to use when a slab being imported does not have a recognized material.

Material Grade

Select the material grade to use when a slab being imported does not have a recognized material grade.

Plate Defaults

Type

By default, all plates are imported as General plate systems. You cannot change this value.

Naming Category

Select a category for the plate system. Categories specify the role of the plate system in the model. The category is also used by the naming rule to name the plate part that is a child to the plate system.

Tightness

Select the level of tightness as it applies to the entire plate system.

Continuity

Select the continuity type for the plate system. Continuity defines how the plate system should react when it intersects another plate or profile system. Select Continuous to indicate that the plate system should penetrate the other system. Select Intercostal to indicate that the plate system should be penetrated by the other system.

Priority

Specify the continuity priority. This priority is used to specify which plate system is penetrated when two plate systems intersect, but have the same Continuity value. Plate systems with a lower continuity priority (1, 2, 3, for example) penetrate plate systems with a higher continuity priority (7, 8, 9, for example).

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Specification

Select the structural specification for the plate system.

Export file

Define the file name and folder path of the CIMsteel Integration Standard file to read. We recommend using a UNC path (\\server\share\filename.stp) in this box if you are going to batch import the file so that the batch computer can locate the file.

Include mapping file

Select this option to use a mapping file when importing the members from the CIS file. You use a mapping file to:

- Swap the third-party software name for a section (for example, L3.5X2.5X1/4) with the Smart 3D name for a section (for example, L3-1/2X2-1/2X1/4). You must create the section mapping file using the **File > New Mapping File Command** before you can use the mapping file in this command. Sample mapping files for the FrameWorks Plus AISC table are delivered with the software in *[Product Folder]*\SmartPlantStructure\Symbols\StructureImportExportMaps. For more information, see *New Mapping File* (on page 535).
- Swap the third-party material and material grade name with the Smart 3D name. A sample material mapping file for AISC is delivered with the software in *[Product Folder]*\SmartPlantStructure\Symbols\StructureImportExportMaps.
- Swap the third-party user-attributes (properties) with the Smart 3D properties names. A sample user attribute mapping file is delivered with the software in *[Product Folder]*\SmartPlantStructure\Symbols\StructureImportExportMaps.

  You must bulkload (create) properties in the Smart 3D catalog for those third-party attributes that you want to map. For example, the third-party application has an attribute called "Expected Service Life" for members. Because Smart 3D does not have a property called "Expected Service Life", you need to add that property to members using reference data bulkload. See the *Reference Data Guide* and the *Catalog User's Guide* for more information on bulkloading.

Mapping file

Specify the mapping file to use if **Include mapping file** option is selected.

Log file

Specify a log file name. You can view the log file after processing by clicking **View Log**.

Action

Select whether to import or preview the contents of the CIS/2 file.

- Select **Import** to import objects from the CIS/2 into the model. This option checks the unique identification numbers in the CIS/2 file with the identification numbers of the model objects finding matching objects. This option allows you to update model objects that had been imported previously. Plate objects in the CIS/2 file are imported as slabs.
- Select **Import Ignore IDs** to import objects from the CIS/2 into the model. This option does not compare identification numbers in the CIS/2 file with the identification numbers of model objects. This option allows you to import the CIS/2 objects as new, unique objects to the model. In marine or material handling modes, plate objects in the CIS/2 file are imported as plates. In plant mode, plates in the CIS/2 file are imported as slabs.
- Select **Preview** to identify problems with the CIS/2 file before the actual import is attempted. Objects in the CIS/2 file that have not been imported cannot be previewed.

Filter

Specifies how you want to preview the CIS/2 file. This option is only available when **Action** is set to **Preview**.

- **Objects in CIS file with unknown sections, material, or type** - Select this option to help identify potential problems with the CIS/2 file before you attempt the import. After using this option, you can add unknown sections, materials, and types to the mapping file.

- **Objects in DB that exist in CIS File** - Select this option to identify objects that are in both the model workspace and the CIS/2 file.
- **Objects in DB that are modified in CIS File** - Select this option to identify objects that are in both the model workspace and in the CIS/2 file but have different attributes (such as section size) in the CIS/2 file. This option is useful when "round tripping" between Smart 3D and another software package when you want to see which objects *were* modified in the other software package.
- **Objects in DB that do not exist in CIS File** - Select this option to highlight objects in the model workspace that do not have a corresponding object in the CIS/2 file. This option is useful when the CIS/2 file was originally exported from Smart 3D and you are "round tripping" between Smart 3D and another software package. Using this option, you can locate members in the model that may have been deleted in the other software package (and therefore, were not exported to the CIS/2 file.) This option is also useful for verifying that all members in the other software package were exported to the CIS/2 file.
- **Objects in DB that are not modified in CIS File** - Select this option to identify objects that are in both the model workspace and in the CIS/2 file that have the same attributes (such as section size). This option is useful when "round tripping" between Smart 3D and another software package when you want to see what objects in the Smart 3D model were *not* modified by the other software package.

Option

Specifies how you want to import objects from the CIS/2 file. This option is only available when **Action** is set to **Import**.

- **Use no defaults** - Select this option to import only those objects known to Smart 3D (known either in the software or in the specified mapping file) sections. Objects with unknown sections are not imported into the model but are noted in the log file.
- **Use defaults if needed** - Select this option to import all objects in the CIS/2 to the model. Objects with sections not found in the software or in the mapping file will be given the sections defined in the **Defaults** dialog box. Objects imported with default sections are noted in the log file.

Submit Job

Activates the **Schedule [Task]** dialog box, which is used to define the batch import of CIS/2 files using SmartPlant Batch Services. For more information on the batch settings, see *Schedule [Task] Dialog Box* (on page 524).

View Log

Displays the import log file. You must click **Apply** when importing in order to view the log at the end of processing or to use the **Preview** option. If you click **OK**, the dialog box is closed at the end of processing and you cannot click **View Log**.

Understanding the Log File

File name : C:\CIMSteel\cis2_out.stp ---- *SHOWS HEADER INFORMATION FROM CIS/2 FILE*
Mapping file used : C:\CIMSteel\AISC_Master_Physical_Map.XML
Intergraph Smart 3D Structure Version : 09.00.10.0003
CIS/2 Version : Tekla Structures Version : Next Build: 4232 Revision : 26.1.2009
File imported on : Tue Jan 19 16:26:34 2010

Parent System : Import
Coordinate System : Global
Importing with : Use no default sections
Processing Options : ---- *SHOWS HOW THE OPTIONS WERE SET ON THE DIALOG BOX*
New : On or Off
    Members : ON
    Slabs and Plates : ON
    Walls : ON
    Openings : ON
    Connections : ON
    Assemblies : ON

Modified : On or Off
    Members : ON
    Slabs and Plates : ON
    Walls : ON
    Openings : ON
    Connections : ON
    Assemblies : ON

**********Summary**********

Linear Members..... 333 ---- *LISTS SUCCESSFULLY IMPORTED OBJECTS TOTAL*
Curved Members..... 0
Slabs..... 0
Walls (imported as Slabs)..... 0
Plates..... 6
Openings..... 48
Stair Assemblies..... 8
    Stair Parts..... 18
Ladder Assemblies..... 3
    Ladder Parts..... 0
Handrail Assemblies..... 76
    Handrail Parts..... 249
Assy Connections..... 0
    Connection Parts..... 0
Embedment Assemblies..... 0
    Embedment Parts..... 0

Total Number of Objects Imported: 741
Total Number of CIS Entities in the import file... 1133
Total Number of Items not found in the catalog..... 360

****

CROSS SECTIONS --- *LISTS ANY MISSING AND MAPPED CROSS-SECTIONS*

Cross Sections	Encountered	Mapped To
PL9.525*127	65	Not Found
W8x13	109	W8*13
W13*56	9	W8x13 (Default)

MATERIALS --- *LISTS ANY MISSING AND MAPPED MATERIALS*

Materials	Encountered	Mapped To
ANTIMATERIAL	30	A36

Import structure completed successfully on: Tue Jan 19 16:29:40 2010.

```
************************DETAILED SUMMARY*********************
Plates Using Default material
2331884
2331863
2331842
1873861
1873822
Linear Members with Missing Cross Sections
2354361
2354229
2013930
2013876
Handrail Parts with Missing Cross Sections
2154615
1995739
1995257
1994775
1994293
Stair Parts with Missing Cross Sections
1908086
1908063
1908040
Zero Length Ladder Parts
2265509
2264060
1880386
Ladder parts with Missing Cross Sections:
1880370
1880231
1880223
1880213
************************************************************
```

Schedule [Task] Dialog Box

Queue

Displays the name of the queues configured by an administrator for the job. For more information on configuring the queues, see *Configure Queues for Jobs* in the *Project Management User's Guide*.

Run job

Sets the frequency with which the job runs. Jobs can be scheduled to run once or on a regular interval (daily, weekly, or monthly). Depending on the job frequency selected, additional controls display. These controls allow you to define more specific scheduling

information. The scheduling controls can be changed only at job submission.

Run on

Sets the time to start running the job.

Options

Opens the *Optional Schedule Properties Dialog Box* (on page 526) that you can use to define a start and end date.

Run on box

Contains a calendar from which you can select the run date. This option is available when you select **Once** from **Run job**.

Every X days

Specifies how many days pass between job runs. This option is available when you select **Daily** from **Run job**.

Every X weeks

Specifies how many weeks pass between job runs. In addition, you can select on which days the job runs. This option is available when you select **Weekly** from **Run job**.

Day X of the month

Specifies on which day of the month the job runs. This option is available when you select **Monthly** from **Run job**.

The X Y of the month

Specifies on which day of the month the job runs. For example, you can select the last Monday of the month. This option is available when you select **Monthly** from **Run job**.

Job Start

Notifies you when the job starts, if Outlook is set up.

Job Completion

Notifies when the job completes, if Outlook is set up.

Job Abort

Notifies you if the job aborts, if Outlook is set up.

Address Book

Selects the name of the person to be notified by e-mail of the job status, if Outlook is set up. If Outlook is not available, this option does not work. You can also type the address manually. The person you define here receives an email with the job log files after the job finishes.

NOTES

- The Batch Services SMTP option must be configured on the batch server for this to work. For more information, see the Intergraph Smart Batch Services documentation.
- The WinZip application is no longer required on the batch server to compress any emailed attachments. Compression is now done with functionality included in Smart 3D.

Optional Schedule Properties Dialog Box

Provides more options on the **Schedule Backup** dialog box. This dialog box opens when you click **Options**.

Start date

Sets an optional start date.

End date

Sets an optional end date, if checked.

Import Bocad

The **File > Import > Structure > Bocad** command imports a Bocad® .bti file into Smart 3D.

Import of Attributes/Properties

Attributes that were defined in bocad and exported can be mapped and imported to Smart 3D properties using the sample UserAttribute_Map.xlm file. These attributes are included in the BOCAD_Master_Physical_Map.xml file. This file is delivered with Smart 3D in the *[Product Folder]*\SmartPlantStructure\Symbols\StructureImportExportMaps folder.

If the bocad attribute that you want to map does not have an equivalent property in Smart 3D, you need to add that property to Smart 3D (using bulkload) before you can map to it in the xml file.

If the bocad attribute does not have an entry in the mapping files but a property with the same name exists in the Smart 3D, then that third-party attribute imports and populates the Smart 3D property even though it is not in the mapping file.

Member Orientation and Member Type

The physical orientation of the member determines the member type during import.

- If the bocad member name exactly matches an existing member type defined in Smart 3D, then the software sets the member to that member type regardless of orientation. For example, if a member has an orientation greater than 10° from vertical but is named "Column", the software imports that member as a column despite the member having an orientation of a vertical brace.
- Members that are vertical or within 10° (1/10 slope) of vertical import as columns.
- Members that are horizontal or within 10° of horizontal import as beams.
- Members that are between 10° of horizontal and 10° of vertical import as vertical braces.
- Members are never imported as horizontal braces as there is no way to distinguish horizontal braces from beams. You must manually edit the members that are horizontal braces after import and set the correct member type.
- All curved members import as beams.

Limitations

- Curved members are imported using the resolution defined in the import file.

- Built-up members are imported if the mapping file is defined to do that.
- Previously imported structure is not updated on subsequent imports. In this situation, either that structure is skipped, replaced, or imported as a copy depending on the import option chosen.

How to Import a bocad File

1. Select **File > Import > Structure > Bocad**.
2. Select the system and coordinate system to use for the imported objects.
3. Select whether to import members (with or without openings) and plates (with or without openings).
4. Define the .bti file to import.
5. Select the mapping file.
6. Define the location and name for the log file.
7. Select the action to perform when you click **Import**.
8. Click **Import**.

bocad Import Dialog Box

Controls how objects are imported from the bocad .bti file. Be sure to check the Compatibility Matrix before you import data. Open *https://smartsupport.intergraph.com https://smartsupport.intergraph.com*, select **View Downloads** at the top, and then select **Product Compatibility** under **Useful Links** on the right.

System

Select the system to which to assign the objects being imported. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model.

Coordinate System

Select the coordinate system to use for the imported objects. All objects are placed relative to this coordinate system.

Members

Select to import members in the import file. Clear this option to not import members in the import file.

Plates

Select to import plates in the import file. Clear this option to not import plates in the import file.

Openings

Select to import opening in members and plates. Clear this option to not import openings.

Import File

Define the file name and folder path of the bocad .bti file to read and import.

Mapping file

Specify the mapping file to use. A sample mapping BOCAD_Master_Physical_Map.xml file is delivered with Smart 3D in the *[Product Folder]*\SmartPlantStructure\Symbols\StructureImportExportMaps folder.

Log file

Specify a log file name.

Action

Select whether to import or preview the contents of the bocad file.

- Select **Import** to import objects from the bocad into the model. This option checks the unique identification numbers in the bocad file with the identification numbers of the model objects finding matching objects. Matching objects from the bocad file are skipped during import because they already exist in the model. Only new members in the bocad file are imported into the model.
- Select **Import as copy** to import objects from the bocad into the model. This option does not compare identification numbers in the bocad file with the identification numbers of model objects. This option allows you to import the bocad objects as new, unique objects to the model.
- Select **Import and replace** to import objects from the bocad into the model. This option checks the unique identification numbers in the bocad file with the identification numbers of the model objects finding matching objects. This option deletes the existing, matching model object and places a new model object from the bocad file. Because of this delete and replace, other objects connected to the deleted and replaced object must be reconnected using *Member Autoconnect* (on page 488).
- Select **Preview** to identify problems with the bocad file before the actual import is attempted. Objects in the bocad file that have not been imported cannot be previewed. All cross-sections, materials, and plate stocks found in the import file are written to the log file.

View Log

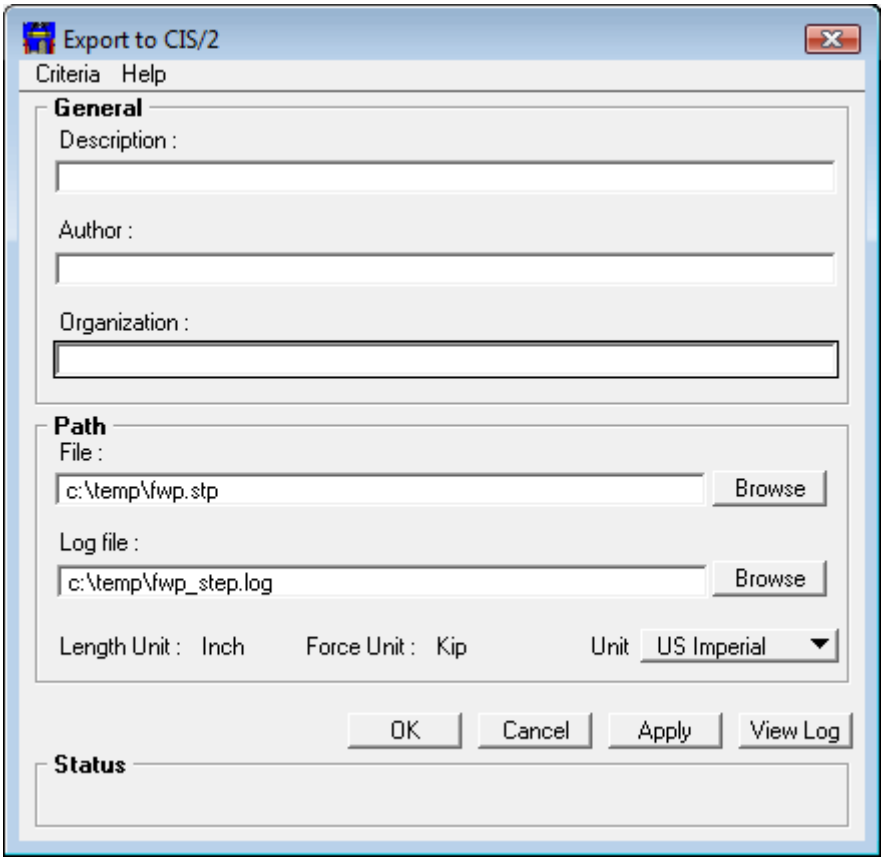
Opens the log file.

Exporting from FrameWorks Plus

The CIS/2 translator for FrameWorks Plus is delivered as part of a FrameWorks Plus. To use this command, open the model that you want to export to Smart 3D. Be sure to attach all reference models that are needed. Then, type **mdl load SPExport** in the MicroStation key-in field.

You can also run the CIM Steel export application, **spexport**, from a command line. For more information, see *CIM Steel Export Key-in* (on page 533).

NOTE FrameWorks Plus does not write generic solid members to the CIS/2 file.



Criteria

Activates a dialog box used to select what members to write to the CIS/2 file. For more information, see *Member Criteria* (on page 531).

Description

Enter a description for the CIS/2 file.

Author

Enter your name.

Organization

Enter your organization name.

File

Enter a name and location for the CIS/2 file.

Log file

Enter a name and location for the log file.

Units

Specify the units for the CIS/2 file.

OK

Writes the CIS/2 file and closes the dialog box. You cannot review the log file using the **View Log** command if you click **OK**. Click **Apply** if you want to review the log file at the end of the process.

Cancel

Exits the dialog box without writing the CIS/2 file.

Apply

Writes the CIS/2 file without closing the dialog box. Use this command if you want to review the log file at the end of the process.

View Log

Opens the log file for review.

Status

Displays the CIS/2 writing progress.

Based-on CIS/2 Statement for Export

Application Name:	Smart 3D
Application Version: Version 2016 (11.0)	**Date:** Monday, September 12, 2016
Translator Version: Version 2016 (11.0)	**Date:** Monday, September 12, 2016
Software Vendor:	Intergraph Process, Power & Marine 300 Intergraph Way Madison, Alabama 35758 U.S.A.

The translators for this application have been implemented in accordance with the second release of the CIMsteel Integration Standards (CIS/2.0) for the following (combination of) Conformance Classes:

- CC312, CC110, CC118, CC255, CC331, CC100, CC308

Type of CIS Translator:	Basic | **DMC** | IDI | PMR-enabled
Data exchange capabilities:	Import | **Export** | Import & Export
Level of implementation:	**File Exchange** | In memory | DBMS | KBS

Flavors supported: EU | **US** | UK

Unit Systems supported: **SI** | **US Imperial**

The vendor places the following riders on the operation of the translators:

- managed_data_deleted, managed_data_creation, and managed_data_transaction are not exported.

Date of Statement: Monday, September 12, 2016

Statement made by: Intergraph

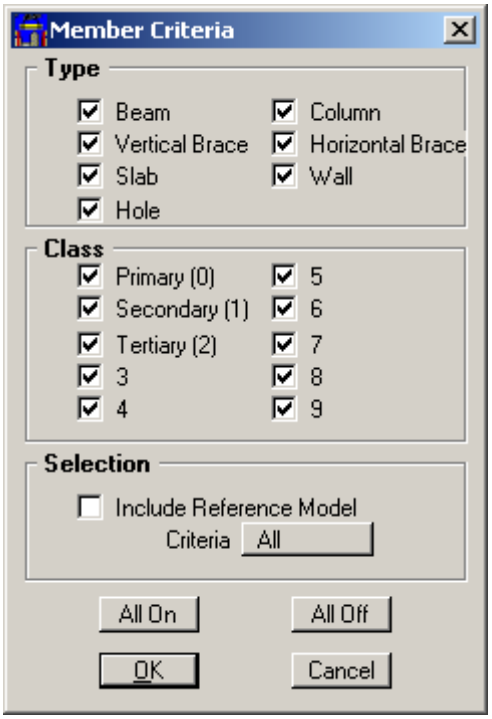
If you have any questions about using this translator, please contact Intergraph support. You can find support information on our web site: http://support.intergraph.com.

See Also

Exporting Models from FrameWorks Plus (on page 532)
Member Criteria (on page 531)
CIM Steel Export Key-in (on page 533)

Member Criteria

The **Criteria > Member Criteria** command defines which members you want to write to the CIS/2 file. Because you define the criteria for what members to write to the file here, the settings in the **Select Filter Settings** command are ignored.



Type

Toggle on the member design types you want to write to the file.

Class

Toggle on the member classes you want to write to the file.

Include Reference Model

Defines whether to write members from attached models to the file in addition to the members in the active model. You can attach models using the **File > Attach Models** command. You can detach models using the **File > Detach Models** command.

Criteria

Defines whether to write members from a selection set or to write all the members in the model to the file.

All On

Toggles on all **Design Type** and **Member Class** settings. The default is that all settings are selected.

All Off

Toggles off all **Design Type** and **Member Class** settings.

OK

Saves any changes you have made and exits the dialog box.

Cancel

Ignores any changes you have made and exits the dialog box.

See Also

Exporting from FrameWorks Plus (on page 529)
Exporting Models from FrameWorks Plus (on page 532)

Exporting Models from FrameWorks Plus

1. Start FrameWorks Plus.
2. Open the model that you want to export.
3. Attach all reference models that are needed.
4. Type **mdl load SPExport** in the MicroStation key-in field.
5. Define the **Criteria** for member selection.
6. Type a description for the file.
7. Type your name and your organization's name.
8. In the **File** box, enter a name and location for the CIS/2 step file.
9. In the **Log file** box, enter a log file name.
10. Select whether to write the SI units or Imperial Units to the file.
11. Click **Apply**.
12. Click **View Log**.

13. Carefully review the log file for errors.

See Also

Exporting from FrameWorks Plus (on page 529)
Member Criteria (on page 531)

CIM Steel Export Key-in

You can export an individual model or all the models in a project using a command line key-in. The command line export model program verifies the validity of each model before processing that model. Invalid models are not processed. Possible invalid models include models from older software versions or corrupted models where a database file is missing.

Error messages are written to a log file created in the user TEMP environment variable folder and is named FWCIMSteelExport.log.

IMPORTANT Before running this command line utility, you must create a text file, inputoptions.txt, which describes all input options based on the following file format.

File Format

```
# Models List
# This section contains a list of models with complete path.

   \\pdsserver\cimsteel\mod\model1
   \\pdsserver\cimsteel\mod\model2

# Project information
# This section includes information about author, organization and
unit.
# For Unit, 1 indicates US_IMPERIAL and 2 indicates SI system of units.

   Author = FWP Developer
   Organization = Intergraph
   Unit = 1

# Member criteria selection for type
# 1 indicates that an item is selected, while 0 not selected.

   Beam = 1
   Column = 0
   VB = 1
   HB = 1
   Slab = 1
   Wall = 1
   Hole = 1

# Member criteria selection for class
# 1 indicates that an item is selected, while 0 not selected.

   Class0 = 1
   Class1 = 0
   Class2 = 1
   Class3 = 1
   Class4 = 1
```

```
Class5 = 0
Class6 = 1
Class7 = 0
Class8 = 0
Class9 = 1
```

The command syntax is:

<prod_dir>\bin\FWCIMSteelExport.exe *<file_path>*\inputoptions.txt

Where <*prod_dir*> is where FrameWorks Plus is installed and <*file_path*> is the path to inputoptions.txt. For example, if FramesWorks Plus is installed under c:\win32app\ingr\fwplus and inputoptions.txt is under d:\temp, then the command is `c:\win32app\ingr\fwplus\bin\FWCIMSteelExport.exe d:\temp\inputoptions.txt`.

This utility creates separate step files for each model and all these step files are created in the project\int folder. The name of the file is the model name appended with _export.stp. For example, if the model name is model1.mod, then the step file name is model1_export.stp.

NOTE You can also run this utility from the PD_shell **CIM Steel Export Batch** command. In this case, PD_Shell creates input options file before running this utility.

Exporting from Tekla Structures

You can export CIS/2 files from Tekla Structures to import into Smart 3D. When exporting from Tekla Structures, we recommend that you setup View Filters in Tekla specifically for isolating only the objects to be exported from Tekla Structures. Also, be sure to select the **Select Object in Components** command to include parts from components in the model, like stairs, in the CIS/2 file.

Be sure to check the Compatibility Matrix before you import or export data. Open *https://smartsupport.intergraph.com https://smartsupport.intergraph.com*, select **View Downloads** at the top, and then select **Product Compatibility** under **Useful Links** on the right.

What do you want to do?

- *Export Structure Model from Tekla Structures* (on page 534)
- *Import Structure Model into Tekla Structures* (on page 535)

Export Structure Model from Tekla Structures

1. Select the parts that you want to export.
2. If you want to include parts from components in the model, select the command.



3. Click **File > Export > CIM Steel > Manufacturing Model.**
4. Select **Properties**.
5. Select the **Parameters** tab.

6. Type a name for the CIM/Steel file in the **Output File** box.
7. Set **CIS/2 Model Type** to **Design**.
8. Select metric or imperial units, depending on your needs.
9. Type the structure name in the **Structure Name** box.
10. Type a path for the profile conversion file, if needed.
11. Type a path for the material conversion file, if needed.

    TIP Smart 3D uses mapping files when importing CIS/2 files, so creating these profile and material conversion files may be redundant.
12. Set **Export globally Unique IDs** to **Yes**.
13. Select whether to include concrete in the CIS/2 file, depending on your needs.
14. Select the **Design Model** tab.
15. Set **Export design connections** to **Yes**.

    TIP This allows Smart 3D to read in the member connectivity of beams, columns, and braces at their work point intersections while still recognizing member cut backs at bracing connections.
16. Save the attribute settings so that you can use them next time.
17. Click **OK**.

Import Structure Model into Tekla Structures

1. Click **File > Import > CIMSteel**.
2. Select **Properties**.
3. Select the **Parameters** tab.
4. Set **Model type** to **Design**.
5. Set **CIS Version** to **CIS/2**.
6. For the **Input file**, browse to and select the CIS/2 file that you exported from Smart 3D.
7. Select the **Conversion** tab, if needed, and browse for the conversion files.
8. Click **Apply**.
9. Click **OK**.

NOTE Tekla has a limit of 21 characters for a part name. If you use a Smart 3D naming rule that creates a part name that is longer than 21 characters, Tekla truncates that name during the CIMSteel import into Tekla. This truncation can cause all your member parts to have the same name in Tekla because Smart 3D naming rules have the unique index at the end of the name.

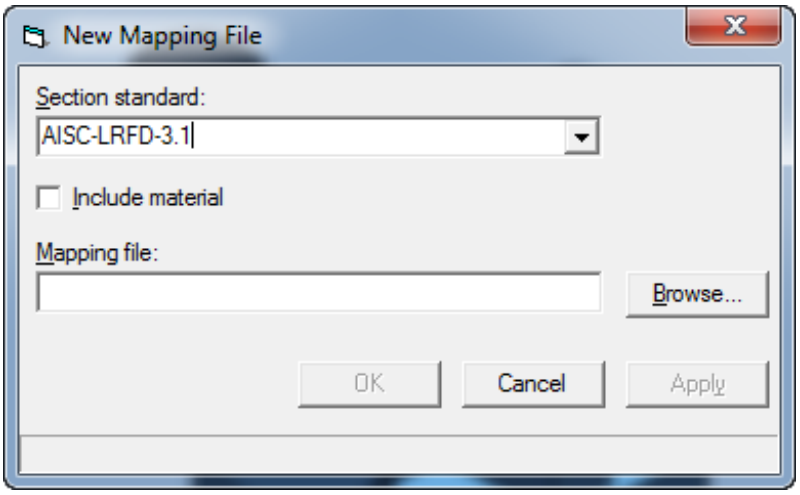
New Mapping File

The **File > New Mapping File** command creates an XML mapping file for the section names, and optionally material names, used in the software and third-party application. Many times, the software and the third-party application use different names for the same section or material. The mapping file solves the naming conflicts by mapping section names in the software to

section names in the other applications. The mapping file must contain each section standard table that you have used in the model.

CAUTION The mapping file created by this command is a template. The software does not write known-to-be-different section names to the mapping file. You are responsible for verifying, editing, updating, and maintaining the third-party application section names in the file.

New Mapping File Dialog Box



Section standard

Select the section standard table for which to create a mapping file.

Include material

Select to include material name mappings in addition to the section name mappings.

Mapping file

Specify a name and folder path for the XML mapping file.

Mapping File Format

The mapping file is an XML-formatted file with which you can define mappings for section names, material names, member types, slab types, and user-defined attribute/properties between Smart 3D and the third-party software. The format for each mapping is given below. You can include all five mappings in a single XML file, or you can define the mappings in separate XML files and reference the five files in a single master XML file using include statements. Using separate files that are pulled together using include statements might be a better workflow as it allows you to quickly mix and match files for different requirements.

You cannot nest include files. Only the master XML mapping file can call an include file. You cannot call another include file inside an include file. An example of a master XML mapping file is shown below:

```
<xml>
<!-- Comment Line -->
    <IncludeXML href="SectionStandard.xml" />
    <IncludeXML href="Material.xml" />
    <IncludeXML href="MemberTypes.xml" />
    <IncludeXML href="SlabTypes.xml" />
    <IncludeXML href="UserAttribute_Map.xml" />
</xml>
```

Section Mapping

The <SectionStandard> area maps the software section names to the third-party section names. The software section names are labeled **section name**. The third-party section names are labeled **externalname**. When the XML file is created, the software section name is duplicated for the third-party section name. You must verify that the correct third-party section name is defined for **externalname** by manually editing the XML file.

```
<xml>
<!-- Comment Line -->
<SectionStandard>
    <Standard name="AISC-LRFD-3.0" externalname="AISC" externalorganization="user"
                  externaldate="2002" externalversion="1.0" />
        <Sections>
            <Section name="W10x39" externalname="W10x39" />
            <Section name="W10x33" externalname="W10x33" />
            <Section name="W10x30" externalname="W10x30" />
            ...
        </Sections>
</SectionStandard>
</xml>
```

Material Mapping

The <MaterialStandard> area maps the software material grade names to the third-party material grade names. You must have selected the **Include material** option when you created the XML file to see the material grade name mappings. The software materials are labeled **Material type** and **grade**. The third-party material names are labeled **externalname**. You must verify that the correct third-party material grade name is defined for **externalname** by manually editing the XML file. You must provide the external material standard name information by manually editing the XML file.

```
<xml>
<!-- Comment Line -->
<MaterialStandard>
    <Standard name="" externalname="" externalorganization="" externaldate="" externalversion="" />
        <Materials>
            <Material type="Steel - Carbon" grade="A36" externalname="A36" />
            <Material type="Steel - Carbon" grade="A529" externalname="A529" />
            <Material type="Steel - Carbon" grade="A588" externalname="A588" />
            ...
        </Materials>
</MaterialStandard>
</xml>
```

Member Type Mapping

The <MemberTypes> area maps the software member types to the third-party member types. You must create this section in a text editor. The software member types are labeled **Member type**. The third-party member types are labeled **externaltype** and **externalrole**.

```
<xml>
<!-- Comment Line -->
<MemberTypes>
    <Member type="Beam" externaltype="Beam" externalrole="" />
    <Member type="Girder" externaltype="Beam" externalrole="gantry_girder" />
    <Member type="Joist" externaltype="Beam" externalrole="joist" />
    ...
</MemberTypes>
</xml>
```

Slab Type Mapping

The <SlabTypes> area maps the software slab types to the third-party slab types. You must create this section in a text editor. The software slab types are labeled **Slab type** and **composition**. The third-party slab types are labeled **externaltype**.

```
<xml>
<!-- Comment Line -->
<SlabTypes>
   <Slab type="4&quot; Cast in Place" composition="CIP_4&quot;_Fc3" externaltype="slab" />
   <Slab type="4&quot; Cast in Place" composition="CIP_4&quot;_Fc4" externaltype="flat_slab" />
   <Slab type="5&quot; Cast in Place" composition="CIP_5&quot;_Fc3" externaltype="wall" />
    ...
</SlabTypes>
</xml>
```

User Attribute Mapping

The <UserAttributes> area maps third-party software attributes to Smart 3D properties. You must create this section in a text editor.

```
<xml>
<!-- Comment Line -->
<UserAttributes>
    <Object type="CSPSMemberSystemLinear" externaltype="assembly_design_structural_member_linear"  role="" >
          <Interface name="IJUAStructuralFrameItemAttributes" externalname="IJUAStructuralFrameItemAttributes" >
              <Attribute name="item_number" externalname="item_number" />
              <Attribute name="item_name" externalname="item_name" />
              <Attribute name="item_description" externalname="item_description" />
              <Attribute name="life_cycle_stage" externalname="life_cycle_stage" />
          </Interface>
          <Interface name="IJUAStructuralFrameProductAttributes" externalname="IJUAStructuralFrameProductAttributes" >
              <Attribute name="life_cycle_stage" externalname="life_cycle_stage" />
          </Interface>

          <Interface name="IJUAAssemblyAttributes" externalname="IJUAAssemblyAttributes" >
              <Attribute name="assembly_sequence_number" externalname="assembly_sequence_number" />
            <Attribute name="complexity_level" externalname="complexity_level" />
          </Interface>

          <Interface name="IJUAADSMAttributes" externalname="IJUAADSMAttributes" >
              <Attribute name="key_member" externalname="key_member" />
              <Attribute name="structural_member_use" externalname="structural_member_use" />
              <Attribute name="Floor Thickness" externalname="Floor Thickness" />
              <Attribute name="structural_member_class" externalname="structural_member_class" />
          </Interface>
    </Object>
...
</UserAttributes>
</xml>
```

Create a Mapping File

1. Click **File > New Mapping File**.
2. In the **Section standard** box, select the section standard for the mapping file.
3. Optionally, select **Include material** to write material names to the mapping file.
4. Click **Browse**, and then specify a name and folder location for the mapping file.
5. Click **OK**.
6. Edit the mapping file using a text editor such, as Notepad, and define the third-party standard section, material names, member types, and slab types.

Glossary

abstract part

A part that is only defined by a partial specification and that cannot be materially provided by the organization that defines the specification.

ACI

American Concrete Institute.

Active Template Library (ATL)

Set of class templates and wizards supplied with Microsoft C++ Version 5.0 and later. You can use an ATL when you create ActiveX controls and any other type of object that uses the Component Object Model (COM) model. Using an ATL is generally preferred over Microsoft Foundation Classes (MFC), because the implementations are smaller, easier to use, and more closely tied to the COM model.

AISC (American Institute of Steel Construction)

An organization responsible for defining American steel construction standards.

AISI

American Iron and Steel Institute

anchor bolts

Bolts used to connect structural members to concrete footings.

angle

The circular measurement taken from the intersection of two pipes at a turn or branch.

anisotropic material

A material that has heterogeneous material properties represented by a material property matrix in three directions.

approval state

Recorded state of acceptance of information contained in objects within the database. The approval states indicate a level of confidence in the information stored in the database and govern your ability to alter specific data about a product.

arc element

A structural member of constant radius used to model curved elements.

arrangement (accommodation)

Those components of a system arranged in three-dimensional space with accurate dimensional representation for installation. Various types include electrical, HVAC, machinery, outfitting, and piping.

arrangement (outfitting)

Outfitting structure arranged in three-dimensional space with accurate dimensional representation.

assembly

Unit composed of a collection of parts or other assemblies. Assembly creation consumes the individual part names and provides the unit a unique identification in the fabrication process.

associativity

A model architecture where the integrity and consistency of the model is guaranteed by the relationships between model entities.

attribute

A single type of non-graphics information that is stored about an object such as diameter or end preparation.

axis

An imaginary line used to define the orientation of a system or object normally defined in terms of an x-, y-, and z-axis. Some 3-D graphic objects have an associated axis used to define the center or axis for rotations.

bar

A steel rod used to reinforce concrete.

basic design

Engineering definition of the model and its systems.

bay

The distance between two trusses.

BCSA (British Constructional Steelwork Association)

An organization responsible for defining British steel construction standards.

beam

A structural member type typically placed with the member axis in a nominal horizontal orientation.

bearing plate

A steel plate used to distribute a load over a larger area. Usually used at the base of a column.

bent

A vertical framework usually consisting of a truss or beam supported at the ends on columns.

bill of material (BOM)

Hierarchical decomposition of a product into constituent assemblies and parts. Specific types of BOMs exist (for example, an EBOM is a bill of material from the point of view of an engineering department; an MBOM is a bill of material from the point of view of manufacturing).

boundary condition

A property that defines the restriction on the allowable direction of movement (degree of freedom) at a particular node.

brace

A diagonal member used to stiffen a framework.

built-up member

A member built from multiple standard shapes to create a single, usually stronger, member.

bulkload

The process by which reference data in Microsoft Excel workbooks is loaded into the Catalog database.

camber

Slight upward curve given to trusses and girders to avoid the effect of sag or self-weight.

cans

A reinforcing connection piece placed at member intersections.

cantilever

A beam, girder, or truss that overhangs one or both supports.

catalog

Repository of information about components and materials used in construction. When you use catalog parts in the model, the software places an occurrence of the catalog part in the project. This occurrence is a copy of the actual catalog part.

Catalog database

The database that contains the reference data. Each model database can reference a different Catalog database.

chain

A set of continuous and tangent segments.

change history

Process of recording information such as who, when, and why for any given modification.

change management

Software features or manual procedures for managing the consequence of change. For example, software can support a change management feature to report drawings that need updating as a result of a change in a 3-D model.

change propagation

Ability of the software to intelligently modify dependent design information to reflect change in a higher order object.

channel

A structural shape referring to a three-sided member type with each of the sides joined at a right angle.

chord

The principal member of a truss. Can be on either the top or bottom of the truss.

circular tube

A structural shape referring to a hollow cylindrical member type or pipe.

CISC (Canadian Institute of Steel Construction)

An organization responsible for defining Canadian steel construction standards.

class

Grouping of individual objects that share some very significant, common characteristics.

classification folder

A folder in the Catalog hierarchy that contains part classes. Classification folders are one level above part classes. The ClassNodeType and R-ClassNodeDescribes sheets in the Microsoft Excel workbooks define the classification folders.

clip angle

A small angle-shaped piece of steel used for fastening members together.

codelist

A set of acceptable values for a particular property that can be referred to by an index number or selected in a combo box. For example, the codelist for the material specification allows you to select from a set of standard entries, such as ASTM A183-F316 Stainless Steel.

column

A vertical structural member usually attached to a footing and extending to the roof of a building.

commodity code

A user-defined code that provides an index to parts in a catalog.

commodity item

A standard component found in a manufacturer catalog (an off-the-shelf component).

component

Physical part that a feature generates.

concurrent access

Ability of the software to allow multiple users to simultaneously access and modify the design of a model.

consolidated tasks

A collection of tasks run in batch. For example, the software allows you to extract a set of drawings immediately or to schedule the batch extraction for a future time.

constraints

A logical restriction that controls how part symbols ports relate to each other and to reference ports. There are four constraints: parallel, perpendicular, coincident, and distance.

construction profile

Principal structural plan for the plant that generally includes a cross section; also an initial structural plan.

contract

A Work Breakdown Structure object representing a scope of work, usually performed by an external supplier. The contract is related to a project and appears in the Work Breakdown Structure hierarchy.

coordinate

The location of a point along the X-, Y-, or Z-axis.

coordinate system

A geometric relation used to denote the location of points in the model. The most common coordinate system is the rectangular coordinate system, whereby points are located by traversing the X-, Y-, and Z-axes of the model. Normally, coordinate systems have their origin defined as 0,0,0.

cope

To cut out the top or bottom flanges and possibly the web so that one member frames into another.

cover plate

A plate used in building up flanges, in a built-up member, to give greater strength and area, or for protection.

cross section

The shape of a member when viewed along the member line.

cutback

An axial offset that typically represents the distance from a member centerline to its face. Cutbacks are used to account for the difference between how a structure is modeled and constructed in terms of lengths of members and quantities of materials.

cutting plane

A plane that cuts through an object.

damage records

Data relating to the damage and repair of structure or components that occurred during or after construction of a plant.

data interchange

Capability to output the design, or portions of the design, in a standard format for use or movement to another computer software system.

database

Repository for the product model data. The database contains information to describe individual objects in the data model and the relationships between objects as appropriate.

database backup

Process of recording a backup copy of the complete database or the incremental changes after the date that the last complete copy was created.

database break and recovery

Utilities used to restore a database after files are corrupted.

database copy

Functionality to copy large collections of model objects from one design project to another design project.

database management

Functionality related to managing a product model database.

database monitor record

Transactions that occur in order to provide database (DB) recovery after a stop in response with a minimum of lost data.

degree

The highest polynomial factor in the curve or surface mathematical definition. A line is a degree 1 curve, while a cubic B-spline is a degree 3 curve.

degree of freedom

An allowable direction of movement, either translation or rotation. There are six possible degrees of freedom (DOFs): translation X, Y, and Z, and rotation RX, RY, and RZ.

design alternative

Difference in a design represented by a separate version. A design alternative can be a new design prepared as a proposed change, or one of several elective options that the builder or customer selects. Each design alternative has an identification assigned so you can uniquely refer to the design alternatives.

design approval log

Record of review and approval of parts of the design.

design data auto input

Automation in loading existing design data into a new design database.

design documents

Drawings, sketches, material lists, procedures, and so forth that are generated during the design phase.

design object

Any object with properties that you can select. A design object can be related to one or more contracts of different types, but related only to one contract of a given type.

design progress check

Analysis of the content of the design to some metric unit that gives an idea of the degree of completion.

design review

Functionality to support rapid viewing of the design and markup of features with comments.

design service

Any general system services related to the design function.

design standard

Feature or object used in plant design that has been determined to the normal or approved way of accomplishing a design requirement. In the context of computer software, the term refers to computer functionality to support standards, not the standard itself.

detail schedule

Lowest level of schedule used to manage and track work progress.

diagonals

A member used for stiffening and wind bracing.

distributed systems

Systems consisting of sequential parts with a distributive characteristic (for example, pipes distribute fluids, HVAC distributes air, cabling distributes power, and structure distributes loads).

distribution systems

Term synonymous and used interchangeably with the term distributed systems.

documentation

Drawings and other records that you must produce to document, obtain approval, or build the design.

drawing tool

Tool that helps in the process of creating, modifying, or manipulating objects. Examples are PinPoint and SmartSketch.

driftpin

A tapered steel pin used to align bolt or rivet holes when assembling steel.

easting

A term that describes an east coordinate location in a coordinate system.

edge

A topological object that represents a trimmed curve bounded by a start and end vertex.

edge distance

The distance from the center of a bolt or rivet to the edge of a plate or flange.

end releases

Physical member properties that define the connection between a member and its nodes. End releases (degrees of freedom) are used to simulate pinned members as well as other special modeling situations.

The member will not contribute stiffness to the node if the degree of freedom (end release) is released. Similarly, the node will not transfer forces or moments to the member through a release degree of freedom. End releases can be defined in any valid degree of freedom for the model.

equipment

Pieces that a foundation supports. Examples are engines, generators, pumps, fans, consoles, large valves, large strainers, and winches. Usually, you can find these pieces on a machinery arrangement plan. Equipment is most often associated with a system.

equipment catalog

Catalog of equipment geometry and limited properties that the software uses to identify and visualize equipment and its placement in the model. The catalog is not the source for the total specification and ordering data for the object.

erection clearance

The amount of space left between members to ease assembly.

fabricate

To cut, punch, and sub-assemble members in the shop.

face-to-face

The overall length of a component from the inlet face to the outlet face.

fasteners

Bolts and rivets used to connect structural members.

element

Primitive geometric shape such as a line, circle, or arc.

fence

Boundary or barrier that separates or closes off an area. To surround or close like a fence.

field adjustment

Material added to the neat design geometry of piping or structural parts to allow for fit up in the case that extra material is required due to uncontrolled variance in the manufacturing and construction process.

finite element

A simple geometric shape defined by a specific number of nodes in a specific order. Elements are dependent on all the nodes defining their shape; if any node is deleted, the element is also deleted. Elements are the building blocks of finite element models. Elements can be one of three types: linear or one-dimensional, plate or two-dimensional, or solid or three-dimensional.

flange

The projecting portion of a beam, channel, or column.

flavor

A different variation of a symbol. Each variation has different occurrence property values.

flexure

A term used for describing bending behavior.

focus of rotation

A point or line about which an object or view turns.

footing

An enlargement at the base of a column, or at the bottom of a wall, that distributes the load over a greater portion of ground and thereby prevents settling.

footprint

A graphical cross-section of a structural member. Footprints are used to represent members that intersect a model view plane at or near a perpendicular angle of intersection.

frame connection

A way of establishing connectivity between member parts to represent physical relationships such as work point offsets, and logical relationships such as end releases. A placement positioning mechanism to represent physical relationships between member systems that establishes and maintains connectivity.

framing plan

A structural drawing plan view, drawn to scale, providing an overhead view of the structural components of a building. Columns, beams, and girders, roof members, floor members, and wall members all require separate framing plans.

Fu

Ultimate strength.

full penetration weld

A type of weld in which the weld material extends through the complete thickness of the components being joined.

function points

Part of the requirements documentation, function points are the smallest granularity of a requirement statement that describe specific detailed actions that the software performs.

functional block diagram

Schematic representation of a system (piping, electrical, ventilation) showing system parts and their relationship. You use symbols to represent equipment and components. A connecting network of lines illustrates their relationship. Taken together, the symbols and the network illustrate the function of the system.

furnishings

Parts such as movable articles and fittings that normally are not associated with a system (for example, a chair).

Fy

Yield strength.

gap element

An analytical element used to simulate one-way compression-only behavior. A gap can be closed and in compression. In addition to the compression, it can then conduct (or not conduct) shearing stress depending on whether or not the gap is sliding. When open, the gap does not participate in the structure stiffness. Typical uses of gap elements in modeling include beams on elastic foundation and the contact of two initially unconnected regions.

generic specific

Object that is parametrically defined or defined to suit a family of specific parts (for example, International Standards parametrics). For example, a 100 - 200 gpm pump in the catalog can provide a general shape to appear in the model until a specific object has been identified. See also specific and specific object.

girder

A horizontal support member similar to a beam. Some people maintain that girders span from column to column, and beams span from girder to girder. Other people maintain that beams span column to column and girders span from beam to beam.

girt

A beam, usually bolted to columns, to support the side covering or to serve as a window lintel.

grade

The material grade of the structural member.

GUIDs

Acronym that stands for Globally Unique Identifiers. The software automatically creates the GUIDs sheet in the Excel workbooks when you create the Catalog database and schema. The purpose of storing GUIDs within Excel workbooks is to help you keep track of what has been loaded into the database. Storing GUIDs also helps to avoid the situation in which a replacement Catalog database causes existing models to become invalid.

gusset plate

A plate used to connect various members, such as in a truss.

hook element

An analytical element or boundary condition used to simulate one-way tension-only behavior. A hook element is a backwards gap with the ability to resist tension only when the hook is closed. It resists neither tension nor compression when it is open. Typical uses of hook elements in modeling include structural bracing, cables, tension, bolts, and the separation of two connected regions.

host location

The first location created for a Site. This host location is defined when the Database Wizard creates the Site database.

host server

The database server on which the Site database was created using the Database Wizard. Alternatively, if it is a restored database set, the Host Server is the database server where the Site database is restored. The Host Server in a Workshare environment contains the origin for the Site, Site Schema, Catalog, and Catalog Schema databases. Consequently, most Project Management and reference data work must take place at the Host.

initial design

Early stage of design work, generally before contract, used to estimate construction costs and provide a rough concept of the intended plant. Contains information relating to a plant created during its initial (concept) design period.

initial structural plan

Principal structural plan for the plant; also called a construction profile.

instantiation

Occurrence of a catalog object at a specific geometric location in the model.

interference checking

A process that identifies possible collisions or insufficient clearance between objects in the model.

I-Section

A structural shape referring to any member type in the form of an I.

isotropic material

A material that behaves with the same material properties in all direction. An example of an isotropic material is steel.

jacket

A three- or four-faced vertical or sloped tower.

job order

Industrial authorization for accomplishing work; synonymous with a work order.

joist

A horizontal structural members that support the floor or roof of a building.

kinematics analysis

Analysis of mechanical motion.

kips

Kilo pounds.

knee brace

A corner brace used to prevent angular movement.

ksi

Kips per square inch.

leg length analysis

Preferred term is welding length analysis.

library

Resource of reference information that you can access in developing a plant design.

life cycle database

Information developed to assist in the maintenance and modernization of delivered plants.

link

Way to store information about another file in your document. You can update a link so that changes in the file appear in your document.

lintel

A horizontal member used to carry a wall over an opening.

load (structure)

A force vector applied to a member.

load group

A grouping in which all components feature uniform load limits and stress safety characteristics. For example, if a pipe clamp from load group 5 has a maximum nominal load of 20kN, then so does a threaded rod from load group 5.

location

A Location is defined by three user-defined inputs: 1) a unique name, 2) a unique name rule ID, and 3) the server where the Site databases reside for that Location. A Location is defined and created when the Site database is created using the Database Wizard. Additional Locations can be created in the Project Management task. Each Location is a Site-level object, thus other Plants within the same Site collection can use the Locations when the Plants are configured for Workshare.

logical member

An object in the model used to represent the design topology.

machinery

Major pieces of equipment installed in a plant.

macro

A sequence of actions or commands that can be named and stored. When you run the macro, the software performs the actions or runs the commands. You can create the macros in Visual Basic or other OLE-aware programming applications. Some of the other OLE-aware programming applications are Visual Basic for Applications, Visual C++, and so forth.

maintenance envelope

A rectangular box around the part for clearance during maintenance operations.

maintenance records

Records of breakdown, repair, and overhaul of equipment.

material analysis

Analysis of a completed design work for extracting detailed material requirements; also called material lists.

material list

An option category that controls the format and content of the bill of materials.

material properties

Properties of the material useful in the analysis process.

member name

A user-definable alphanumeric code used to uniquely identify individual members in the model.

member part

A model object derived from the logical model that represents the manufactured physical member parts.

member system

A logical collection of member parts that can be moved as a single entity.

methods

Objects in the database that describe the manufacturing methods to the component parts of a plant.

move from point

Starting point for an action. For example, when you move an equipment object, the Move From point determines the point of origin for the move.

move to point

Ending point for an action. For example, when you move an equipment object, the Move To point determines where you want the move to stop.

MTO neutral file

A non-graphic output file that can be fed into a material control system. MTO stands for Material Take-Off.

node

- One of the set of discrete points in a flow graph.
- A terminal of any branch of a network or a terminal common to two or more branches of a network.
- An end point of any branch or a network or graph, or a junction common to two or more branches.

northing

A term that describes a north coordinate location in a coordinate system.

nozzle

A piping connection point to a piece of equipment.

nozzle standout

The shortest allowable distance between the connection point of a nozzle and the start point of a turn on the leg connected to the nozzle.

NPD (Nominal Piping Diameter)

The diameter of a pipe.

object

A type of data other than the native graphic format of the application.

occurrence (of part or equipment)

Instantiation of a part of equipment in the model that refers to the part library; an instance of a specific object. The design can be built several times, and therefore the occurrence can apply to more than one hull. Typically, an occurrence points back to a specific object, either for its complete definition, as in the case of a particular valve, or for its made from material, as in the case of a steel plate part cut from sheets. Thus, when a designer selects a component from the catalog and places it at a location in the space of the plant, the software creates an occurrence of that object in the plant design.

occurrence property

A characteristic that applies to an individual object in the model. Occurrence properties are designated with 'oa:' in the reference data workbooks. You can view and modify occurrence properties on the Occurrence tab of the properties dialog boxes in the software. Depending on the object, some occurrence properties are read-only.

orientation vector

A vector used to define the local y-x plane of a physical member when combined with the local x-axis defined by the start and end nodes.

origin

In coordinate geometry, the point where the X-, Y-, and Z-axes intersect.

origin point

The point at which the coordinate system is placed, providing a full Cartesian coordinate system with positive and negative quadrants. Points are placed at coordinates relative to the origin point, represented by the X, Y, and Z values.

orthogonal

The characteristic of an element consisting completely of elements positioned at 90-degree angles. A square is an orthogonal element.

orthographic

A depiction of an object created by projecting its features onto a plane along lines perpendicular to the plane.

orthotropic material

A material that has two material directions that are orthogonal to one another. An example of an orthotropic material is wood.

P&ID

Diagram that shows the topology, functional components, and special requirements of a piping system; generally represents the engineering design of the system.

package

Set of closely related classes. (UML)

painting

Computation of paint surface and recording of paint system requirements.

panel

The space between adjacent floor supports, or purlins, in a roof.

parameter

A property whose value determines the characteristics or behavior of something.

part class

A group of similar objects. You can define part classes in the Excel workbooks. A part class can have multiple parts. For example, a heat exchanger part class can contain heat exchangers with different dimensions.

part number

Unique identifier of a part.

PDS (Plant Design System)

A comprehensive, intelligent, computer-aided design and engineering application for the process, power, and marine industries. PDS consists of integrated 2-D and 3-D modules that correspond to engineering tasks in the design workflow.

physical member

A concept that helps to eliminate modeling errors by providing a straightforward method for defining structure geometry by placing members in a model in much the same way that they would be framed in the field. During analysis pre-processing, physical members are split into finite elements (segments) while still maintaining the collective status of the physical member during model revisions. The software assigns member releases at each end of the true physical member, while preventing release assignments from being made to interior nodes of the finite elements.

PinPoint

Tool that allows you to place, move, and modify elements with precision, relative to a reference point.

pitch

The ratio of rise to run for roofs. Pitch is also the center distance between bolts or rivets, parallel to the axis of the member.

plate

A flat, rectangular steel shape.

post tensioning

A method of pre-stressing concrete, by stressing the steel strands after the concrete has been poured and allowed to harden.

precast concrete

Concrete members that are poured in forms at a plant or factory and allowed to harden. Two types of precast products exist: pre-stressed products and reinforced products.

pre-stressed concrete

Concrete products that are stressed by passing high-strength steel strands through the form and applying stress to the strands either before or after the concrete is poured.

pretensioning

Stressing the steel strands in a pre-stressed member before the concrete is poured into the form.

primary member

A main structural support member.

principle of superposition

The principle that states that the stresses, strains, and displacements due to different forces can be combined. This principle is only valid for linear analysis.

Product Data Management (PDM) System

Software intended to manage both product data and documents associated to the product data. Functionality typically includes: object-based data modeling tools, user administration, business rules, and document management. Document management typically includes document editing or reviewing, document mark-up or redline, document storage, and full-text retrieval.

product structure

Hierarchical breakdown or decomposition of a product into constituent parts, volumes, or units. (For example, a bill of material is one possible type of product structure.)

production planning

Functionality associated with the work breakdown and sequence of the construction of a plant.

promotion

Process of associating approval state with a product version. A product version begins its existence at a working approval state. When the version is at some level of maturity, its approval state is elevated to a higher approval state (that is, promoted). Then, further changes must be carefully controlled and generally require the data set demoted to a working state. One or more promotions can occur successively higher approval states (between working and approved) to represent various intermediate levels of review or progressive approval.

purlin

Horizontal structural member extending between trusses, used as beams for supporting the roof.

query select sets

Set of objects that are selected in a query or queries on the database.

rafters

Beams or truss members that support the purlins.

rebar

A term for steel reinforcing bars that are used to reinforce concrete.

rectangular bar

A structural shape referring to a four-sided solid bar.

rectangular tube

A structural shape referring to a four-sided hollow member type.

reference data

The data that is necessary to design plants or ships using the software. Reference data includes graphical information, such as symbols. It also contains tabular information, such as physical dimensions and piping specifications.

reflect

A parameter that affects the placement orientation of member cross sections. At your option, the sections can be placed with their geometry reflected or mirrored about the local section y-axis.

reinforced concrete plan

A framing plan that identifies all beams and slabs by letter and number. The slab, beam, and bend schedules give all details for preparation and placement (number, size, spacing, bending, and location) of the reinforcing steel.

resource estimation

Rough estimate of material, manpower, and facility utilization for the design and construction of the plant.

retaining wall

A structural wall, either gravity or cantilever, used to hold back dirt or other materials.

route

1) A line connecting a series of points in space and constituting a proposed or traveled route. 2) The set of links and junctions joined in series to establish a connection.

rule-based joint

A feature that enables you to offset the work point of two members that intersect at a node. The rules vary depending on the structural type, member orientation, and construction practices. Ruled joints are typically used when a connection detail calls for the physical member ends to be noncoincident with the centerline of an intersection member.

sag ties

Tie rods between purlins in the plane of a roof, used to carry the component of the roof load parallel to the roof. Tie rods are used to support girts.

SAISC (South African Institute of Steel Construction)

An organization responsible for defining South African steel construction standards.

satellite server

The database server where the replicated databases reside for Workshare. The Satellite Server is not used unless Workshare is activated.

schema

A database that creates the structure of another database. For example, a schema specifies the queries, tables, fields, and data types in a database.

schema update utility

Functionality used to assist in processing existing product models to an updated database structure after you modify or add to the database structure.

secondary member

A member that is not a significant load-bearing member.

section

A structural member whose parameters are defined in a table.

section name

An alphanumeric code used to refer to a particular member in a table; for example W18X35.

section orientation

The position of physical member cress section relative to the physical member local coordinate system.

section table

A library of standard structural shapes containing the necessary parametric properties of each section size, such as depth, width, and inertias.

shear stiffness

A physical member property that indicates whether to include shear stiffness in the analytical element stiffness matrix of a member.

shear walls

A wall designed to resist lateral loading from winds, underground disturbances, or blasts.

site

The top level in the Project Management hierarchy. A Site configuration may contain several Catalogs, each shared by multiple Plants.

site administrator

Person responsible for managing the standards and general parameters for a given plant site within a Site database.

site setup

Functionality associated with establishing a new plant site or hull for design development.

sketch and trace

User interface for rough definition of a required design feature that typically works in a 2-D mode.

skewed member

A structural member that is not at right angles to its connecting members.

slab

A flat concrete area usually reinforced with wire mesh and rebar.

span

A group of contiguous physical members that are not intersected by a brace- or column-type physical member.

specifications

Contracted requirements for the plant.

splice (structure)

To fasten together two physical members, usually columns end-to-end, to form a single continuing length.

steel reinforced concrete

An association of concrete and steel where the concrete is there to stand the compression forces and the steel the traction forces. The resulting structure can withstand traction and flexion.

stiffener

An angle, plate, or channel fastened to a member to prevent buckling.

stress

Forces acting on structural members due to various types of loads. These forces can be shear, tension, compression, or torsion.

stringer

A longitudinal member used to support loads directly.

structure analysis

Analysis routines that provide stress and deflection data for structural designs. Loading conditions can be both static and dynamic. Finite element analysis is the most common type of structure analysis.

strut

A compression member in a framework.

stud

A bolt, threaded on both ends, used to connect components.

suspended floor

A concrete floor system built above and off the ground.

symmetric node

Type of vertex on a curve. A curve with a symmetric node has the same curvature on each side of the node. A handle can be attached to a symmetric node for editing.

system

A conceptual design grouping that organizes parts in hierarchical relationships. A system represents a functional view of the model and includes information such as system name, type, properties, and design specifications for the objects assigned to the system.

tag number

User-specific, unique number assigned to an object (for example, CV-101 for a control valve, HE-2002 for a heat exchanger).

target point

The origin for coordinate measurements displayed by PinPoint. You can position the target point anywhere on the drawing sheet or view.

tilt-up walls

Concrete walls that are poured in forms on the ground and then tilted up into place by cranes or hoists.

tolerant geometry

A type of ACIS geometry - either an edge or a vertex - that is outside the tolerance for ACIS and requires special handling.

transition

A cross-sectional type that results in a tapered length of member and involves a gradual change in section size. See also uniform.

trimmed surface

A surface whose boundary is fully or partially inside the "natural" geometric definition of the surface. Some or the entire control polygon extends outside the face boundary.

trunk

Feature that quickly reserves space for the distributive systems and other systems that have a path. Along the trunk are stations that define the cross section and identify part or system membership.

truss

A rigid framed structure consisting of straight members joined to form a pattern of interconnecting triangles for carrying loads.

uniform

A cross-section type that results in a uniform length of member and involves an abrupt change in section size. See also transition.

unit/module modeler

Facility of the system to structure collections of equipment and components into a single identifiable object.

user attributes

A customized property in the reference data. The Custom Interfaces sheets in the Excel workbooks define these properties. You can list the customized properties on the individual part class sheets.

version control

Ability of the system to manage multiple versions of a single part of the design. Version control should support conditional analysis and promotion status, as well as alternate design features among hulls within a plant site.

vertex

A topological object that represents a point in the three-dimensional model.

viewset

Set of objects (usually a subset of the entire database) that a view operation uses. Membership or lack of membership for any object in a viewset does not affect the actual stored representation of the object, but only its availability or desirability for viewing in the current scenario.

web

The center section separating the flanges of an I-section, tee, or channel.

weight and CG analysis

Routines that compute the weight of commodity materials as configured in a given design (for example, plate and pipe) and determine total weight and center of gravity (CG) for a collection of material and equipment, as well as the complete plant.

welding

Weld requirements for joining materials. Welding length analysis is the calculation of required weld dimensions; also called leg length analysis.

wirebody

A topological object that represents a collection of edges jointed at their common endpoints.

wizard

Software routine attached to an application that provides guidance and expert help to you to complete one of the functionalities of the application.

work content

Estimation development of metrics from the database that relates to the work hour content of the various construction units.

work order

Plant authorization for completing work; synonymous with a job order.

working plane

The available 2-D plane of movement for endpoint selection.

workset

Set of objects (usually a subset of the entire database) used in an interactive change, add, or delete operation. Membership or lack of membership for any object in a workset does not necessarily affect the actual stored representation of an object. However, you can change or delete an object in a workset that also results in a change or deletion of the stored object. Similarly, when you add a new object (not currently stored) to a workset, the software also adds the object container.

workspace

Area that represents the portion of the model data needed to perform the intended task and includes the user modeling settings.

workspace document

Document into which you can extract a portion of the model data for a user task.

Workspace Explorer

Tree or list representation of objects in your workspace.

X-section

Cross section; a graphically placed cross section representing the member section size.

yield strength

The stress beyond which a material sustains permanent deformation.

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